uc3m | Universidad **Carlos III** de Madrid

University Degree in Aerospace Engineering

2017-2018

Final Degree Project

"Numerical analysis of a blast inside aircraft fuselages"

Rocío Fontecha Ballesteros

Tutors

Jose Alfonso Artero

Jesús Pernas

Mechanics of Continuos Media and the Theory of Structures Department

Leganés, October 2018

SUMMARY

The main objective of this Final Degree Project is the structural analysis of the surface or floor, on which the passengers of an aircraft will be during the flight, to search for the preservation of its structural integrity, after having been loaded with the explosion of a certain TNT mass.

To make possible the analysis, it is necessary to implement the Finite Element Method (FEM) not only for the definition of the fuselage model but also for the definition of the blast, throught the CONWEP method.

The conservation of the structural integrity of the passengers' cabin floor is based in the avoidance of the plastic deformation of the structure caused by the stresses, adding to the analysis displacements and pressures, after the detonation.

Three configurations are going to be implement: Cockpit, Engine or Lateral and Hold configurations, in order to determine which is the most critical for the integrity. The TNT mass will be also varied to achieve the objective.

Key words: Blast, displacement, stress, pressure, structural integrity, plastic deformation, cockpit, engine, hold, FEM and CONWEP

DEDICATORY

Primero, me gustaría agradecer a mis tutores de proyecto, José Alfonso Artero y Jesús Pernas, por todo el apoyo que he recibido, al igual que a todo el equipo docente, tanto de la Universidad Carlos III de Madrid como de la Universidad de Sevilla, por los cuales he podido realizar este Proyecto de Fin de Grado.

GRACIAS a mis padres, por todo el apoyo que he recibido durante estos largos 5 años, cada uno a su manera entre gritos y 'Vete a comprar a Zara para despejarte' o las frases filosóficas sobre posibles futuros problemas a los cuales yo ya había buscado millones de soluciones. Otro GRACIAS a mi hermano Nacho, Pi para la familia, que aunque hayamos estado 4 años separados, con este último creo que ya he recuperado todos los 'Dame un abrazo/beso que soy tu hermano preferido'. Muchísimas gracias de verdad.

No me puedo olvidar de mis abuelos, Celsi y Rosarito, por todo su apoyo desde el principio y todas las velas puestas en la Iglesia para que aprobase los exámenes.

A mis amigos de Madrid, con los que comencé la etapa que cierro con este proyecto, tanto de la residencia como de clase, sobre todo a Isa y Celia, Por todos los ratos de risa, llando, conversaciones filosóficas y no tanto, salidas, fiestas, etc. Espero que, aunque este último año hayamos estado cada una en una punta, todo vuelva a ser como antes.

Personas que ya no forman parte de mi vida, como lo habían hecho hasta hace no tanto, pero que han sido muy importantes en este periodo de tiempo. Gracias no solo por el apoyo que siempre he recibido sino también por haberme aguantado, que se que no es poco.

No me puedo olvidar de la última incorporación a mi vida, los Pmoñeros, gracias por haberme acogido este último año de carrera, por haberme dado la mejor excursión de fin de carrera y haberme convencido de que Sevilla es la ciudad donde mejor se vive y sin olvidarme de su Feria. Sobre todo a Ángel, Inma y Marta.

Con este proyecto cierro un ciclo con el que he podido crecer en todos los ámbitos de mi vida.

CONTENTS

1. INTRODUCTION . . . 1
1.1. Motivation . . . 1
1.2. Objective . . . 4
1.3. Contents . . . 4
2. STATE OF THE ART . . . 5
2.1. Aluminum and alloys . . . 5
2.2. Finite Element Method, FEM . . . 6
2.3. Blasts and Conwep method . . . 8
2.4. *Abaqus* software . . . 9
3. NUMERICAL MODEL . . . 11
3.1. Introduction . . . 11
3.2. Development of the model . . . 11
3.2.1. *Abaqus* Modules . . . 13
3.3. Failure Criteria . . . 26
4. ANALYSIS AND RESULTS . . . 27
4.1. Introduction . . . 27
4.2. Simulations . . . 28
4.2.1. Detonation of 1 [Kg] of TNT . . . 28
4.2.2. Detonation of 4.5 [Kg] of TNT . . . 34
4.2.3. Plastic Deformation in the Cockpit and Engine/Lateral Configurations . . . 41
4.3. Evolution of the results . . . 42
5. CONCLUSION . . . 46
5.1. Future Developments . . . 46
BIBLIOGRAFíA . . . 47

LIST OF FIGURES

1.1 September 11 World Trade Center attack. Serra, Alfredo (2016). Photography credit: Infobae 2

2.1 Material distribution in the B787. Photography credit: Aviationknowledge 5

2.2 Example of a mesh in an aircraft. Sevilla, Rubén. (2014). 8

3.1 Real passengers cabin, Photography credit: Havkar 11

3.2 Fuselage skin. Photography credit: OGGturbojets 12

3.3 Set of frames in a fuselage. Photography credit: Boeing 12

3.4 Set of vertical stiffeners. Photography credit: CanStockPhoto 12

3.5 Floor of the passengers cabin. Photography credit: Avitation 13

3.6 Fuselage skin 14

3.7 Upper surface 15

3.8 Lower surface 15

3.9 Frame 16

3.10 Vertical stiffener 16

3.11 Final assembly of the fuselage 19

3.12 Tie interaction between components 21

3.13 Boundary condition 22

3.14 Skin mesh 23

3.15 Upper surface mesh 23

3.16 Lower surface mesh 24

3.17 Frame mesh 24

3.18 Vertical stiffener mesh 25

3.19 Final fuselage mesh 25

3.20 Stress-Strain Curve. Photography credit: Instructables 26

4.1 Location of the detonation points related to the target surface 28

4.2 Vertical displacements for 1 Kg of TNT in the Cockpit configuration . . . 29

4.3 Stress concentration for 1 Kg of TNT in the Cockpit configuration 29

4.4 Pressure field for 1 Kg of TNT in the Cockpit configuration 30
4.5 Vertical displacement for 1 Kg of TNT in the Engine configuration 31
4.6 Stress concentration for 1 Kg of TNT in the Engine configuration 32
4.7 Pressure field for 1 Kg of TNT in the Engine configuration 32
4.8 Vertical displacement for 1 Kg of TNT in the Hold configuration 33
4.9 Stress concentration for 1 Kg of TNT in the Hold configuration 33
4.10 Pressure field of the structure for 1 Kg of TNT in the hold 34
4.11 Vertical displacement for 4.5 Kg of TNT in the Cockpit configuration . . 35
4.12 Stress concentration for 4.5 Kg of TNT in the Cockpit configuration . . . 35
4.13 Pressure field for a 4.5 Kg of TNT in the Cockpit configuration 36
4.14 Vertical displacement for 4.5 Kg of TNT in the Engine configuration . . . 37
4.15 Stress concentration for 4.5 Kg of TNT in the Engine configuration 38
4.16 Pressure field for 4.5 Kg of TNT in the Engine configuration 38
4.17 Vertical displacement for 4.5 Kg of TNT in the Hold configuration 39
4.18 Stress results for 4.5 Kg of TNT in the hold 40
4.19 Pressure field for 4.5 Kg of TNT in the Hold configuration 41
4.20 Plastic deformation in the Cockpit configuration 42
4.21 Plastic deformation in the Engine/Lateral configuration 42
4.22 Evolution of the stresses with time 43
4.23 Evolution of the displacement with time 44
4.24 Evolution of stresses with TNT mass 45
4.25 Evolution of the displacements with TNT mass 45

LIST OF TABLES

2.1 Properties of AL 2040-T3. UNE 38-0001-85 6

3.1 Components of the aircraft 17

3.2 Density and elastic properties 17

3.3 Plastic and Damage properties 19

3.4 Coordinates of Reference Points 21

1. INTRODUCTION

In this first chapter, the project will be introduced to the reader, explaining which are the main causes that leaded to develop this complex nature problem. This introduction is included in the Motivation Section. After setting the historical frame of the project, the principal objectives of the analysis will be described in the Objective Section. Finally, a short description of how is going to be divided the project will be explained in the last section of this chapter.

1.1. Motivation

There is a growing interest on how a structure behaves after have been loaded with a blast due to the importance of preserve the structural integrity, to avoid any kind of damage in people being inside the structure or in the surroundings.

In the last years, the aeronautical industry has been involved in many terrorist attacks, due to the fact that terrorist groups are always looking for damaging the biggest quantity of people. These groups found new ways to introduce exploives into commercial aircrafts, like body implants, but they also continued using other techniques such as explosives inside the luggage or between clothes. [1]

One example of this type of terrorism was the an American Airlines B767 from Paris to Miami. In this case the terrorist introduced the explosive device hidden in the shoes heel. Fortunatelly, passengers noticed the device and they could stop him before the detonation of the device. [2].

Passengers from the Avianca flight 203, from Bogotá to Palmira, did not have the same fortune as the aircraft exploded during the flight. This terrorist attack was associated to the Cartel de Medellín. [3].

They are not the only tragic events that involved the aeronautic industry. The tragic event that change the world, the terrorist attack in New York against the World Trade Center, used two aircrafts as the mean of the attack. Although nowadays facts like arrive to the airport minimum two hours in advance, going through several security controls or taking out of the suitcase any type of electronic device for their scan, they were imposed after this fatal day.

Fig. 1.1. September 11 World Trade Center attack. Serra, Alfredo (2016). Photography credit: Infobae

In terms of security measures introduced at the aircraft design after the World Trade Center attack, the cockpit door was armored in order to avoid the hijack of the aircraft by passengers, numerical codes to close that armored door or not allowing the entry to anyone other than the pilots in the cockpit. ([4]).

Years later, these measures turned against the aeronautical industry with the Germanwings tragedy. The copilot hijacked the aircraft, closing the armored door, when the pilot left his position to go to the bathroom. This action ended with the intentionally crush of the aircraft into the french Alps.

Because of this fact, new security measures were implemented, such as there must be always two people inside the cockpit or increase the health tests to pilots and copilots.([5]).

Not only these intentional events were taking into account in order to develop this Final Degree Project, but also the fact that any kind of electronic device installed in the aricraft or carried by any passenger can explode and caused damages on passengers.

All these intentional or accidentally events have made us notice the vulnerability of aircraft, affecting the integrity of the structure and so people inside it. Hence, it is necessary to look for solutions, that increase the security of the air transport, through the analysis of these types of situations, that can cause the loss of lifes, in a reliable simulation software.

Before the appearance of numerical models, solutions for the situations explained before, were looked for through experiments. Although experiments produced a very reliable database, these supposed a great economic and time investment, with the addition of a great quantity of lost material.

Thanks to the appearance of the numerical models and the technologies advances, experiments were left aside, to use a more efficient tool: the numerical simulations. These simulations supposed the obtaining reliable results in a shorter period of time and the fact that any configuration or situation imagined can be virtual tested, before the implementation of solutions in aircrafts.

Related with the type of analysis that is going to be developed in this project, it can be said that blast problems have a complex nature, due to its dependance with time, meaning that it is a non-linear analysis. The complexity of the problem is also due to the fact that they are applied to structures such as buildings or, like in this project, aircrafts with complicated geometries and big dimensions.

The project is the result of the need for research on the preservation of an aircraft structural integrity and so the passengers protection when events like the ones named before take place.

1.2. Objective

The main goals of this project are going to be introduced in the following points:

- Design numerical application in the simulation software *Abaqus* of the central structure of an medium size commerical aircraft, focused on the cabin in which passengers stay during the flight. For the numerical modelling, the Finite Element Method (FEM) is going to be implemented.
- The load applied will be a explosive one and it will be modeled using the CONWEP Method. To analyze the effect of the blast on the fuselage, its mass as well as the focal point of the detonation will vary.
- The analysis will be based on the results of stresses, displacements and pressures. After obtaining the results, they will be compared to each other to obtained which is the most critical point of detonation, between the three selected, in order to reach the plastic deformation of the structure.

1.3. Contents

This Final Degree Project starts with a short introduction to the aeronautical industry and the objectives to achieve. An explanation of every method applied, to analyze the blast inside the aircraft fuselage, is also included.

Finally, all the results will be commented and compared to each other, to determine which is the most critical position to detonate to achieve the plastic deformation of the structure, that affects to the structural integrity of the aircraft.

2. STATE OF THE ART

In this second chapter, there will be an introduction to the Aluminum and its alloys, as it is going to be applied to the aircraft fuselage in these sets of simulations. In order to obtain results numerically, the Finite Element Method (FEM) is explained, as it is the based of any Mechanics Continuous Media problem. Once the material and the FEM is defined, the blast is defined with the implementation of the CONWEP method, based on the Finite Element Method (FEM). Finally, a short introduction to the simulation software selected for developing this project, *Abaqus/CAE*, will be included.

2.1. Aluminum and alloys

In the beginnings of the aeronautical industry and most of the past century, the material most applied in aircrafts were the Aluminum and its alloys. Nowadays, although composites are gaining ground to Aluminum, this last one remains at sections were is likely to be impacted, like the leading edge of the wings ot the vertical stabilizer. An example of the distribution of materials applied in the B747 is shown in the picture (2.1):

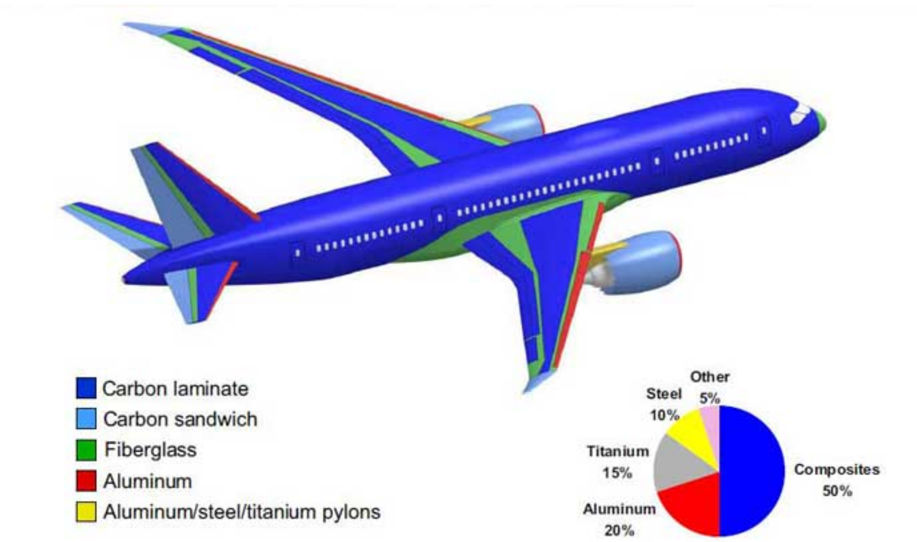


Fig. 2.1. Material distribution in the B787. Photography credit: Aviationknowledge

These materials are not only used due to its high resistance but also due to its values for the density, although nowadays composite materials have lowers values for the density. These properties are very significant for the aeronautical industry, as one of the main objectives is the reduction in weight of the aircraft. This reduction in weight means less fuel consumed and greater economical savings for companies. [6]

Other features, that can be added to de Aluminum, are that can be easily machined and welded or that it is stainless, a very important property, as the aircraft is always exposed to atmospheric changes, that can turn to storms and rain. As desadvantages, this material can have fatigue problems or the fact that the process to obtain it has a high cost.

In this project, the material applied to the aircraft will be the Aluminum alloy AL2024-T3. The nomenclature is explained in the table (2.1):

Digit	**Meaning**
2xxx	Aluminum-Copper
x0xx	0 changes in original alloy
xx24	Purity of Aluminum
T3	Treatment

Table 2.1. PROPERTIES OF AL 2040-T3. UNE 38-0001-85

The last two digits, T3 means that the alloy has a heat treatment following hot working, quenching, cold working and being naturally aged to a stable condition.

In the following chapter, the properties implemented in Abaqus as well as the Johnson-Cook criterion for the plastic deformation of the material are going to be explained.

2.2. Finite Element Method, FEM

In the beginning of the industry, when it comes to solve any kinf of engineering problem related to the Mechanics of Continuous Media and the Theory of Structures, they were almost impossible due to the complex nature of the equations goberning these problems.

This fact made engineers invest a lot of time solving manually these equations and, Universities and companies invest large economical quantities to implement small changes and improvements in experiments to create a database to, later, introduce these changes in the design level of aircrafts.

Thanks to the appearance of the FEM and the technological advance, the problems named before can be solved in a much shorter period of time, with a great accuracy. This method is nowadays one of the most important tool in order to solve Mechanics Continuos Media problems in any engineering sector.

The application of the FEM consist on the discretization or division of an structure in a finite number of elements, with a much smaller size than the original structure, and interconnected between each other through points called nodes. Hence, the elements will act like Degrees Of Freedom.

Besides this elements are smaller than the original one and it is easier to work with them, they maintain all the properties, characteristics and equations that govern the principal structure. The problem will be solved through approximations to the real case, depending on the elements or degree of freedoms in which is divided.

To implement the FEM in any simulation software, first, the definition of the structure geometry is needed for the analysis. The design can be done in the Design Module if the simulation software choosen has it or use a Design software like *SolidWorks* or *CATIA* so as to create a CAD (Computer-Aided Design) file to import to.

Continuing with this resolution method, the introduction of boundary conditions is also relevant to finally get the results. As boundary conditions can be understood as any known parameter than can cause changes in the results of the problem. For instance, a boundary condition can be the displacement and/or rotation restriction.

Finally, the variables, that are going to be analyze, have to be defined. These variables are defined for each of the nodes explained before and they can be stresses, pressure, displacement, etc.

The resolution process starts with the application of the selected type of load on the structure, that in this project the load applied will be the blast caused by certain TNT mass. The load will cause movements or unitary displacements between nodes that connects the elements. Once, these displacements are known, the previously defined parameters or the problems unknowns can be calculated. [7]

Notice, that as it has been said before, the solution of the problem is an approximation to the real case. For this reason, the smaller the elements in which the structure is divided the most realiable results can be obtained. Also, the geometry has to be taken into account, because the resultant mesh in a flat plate will not be the same as a mesh associated to a curved surface. The elements will have different shapes to fit correctly to the origional shape.

In the image (2.2), an aircraft meshed is shown in order to observe the element shapes explained previously:

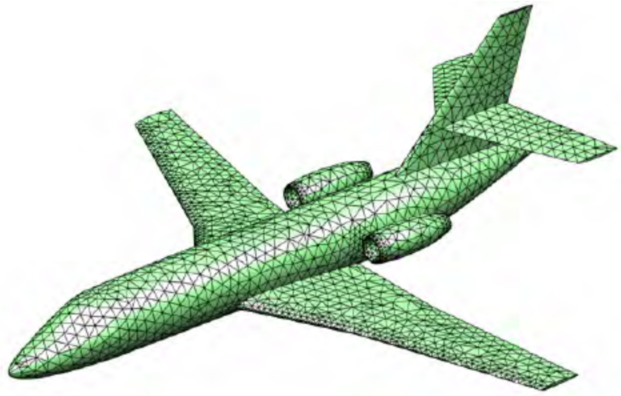


Fig. 2.2. Example of a mesh in an aircraft. Sevilla, Rubén. (2014).

2.3. Blasts and Conwep method

Considering the problems explained in the Motivation Section, explosions has aroused the interest in how structure behaves under this type of loads. In this section, the evolution of a blast, how it affects a structure and how is numerical implemented in a simulation software are going to be explained.

After the detonation of an explosive mass, a pressure wave is produced with a finite amplitude due to the freeing of a huge quantity of energy in a very short period of time, therefore the analysis implemented must be a non-lineal one. When the blast wave reaches the target structure, it produced a peak of overpressure in that point and as it propagates along the structure, the pressure field will variate with time. This fact causes that in the same surface, we can have compressive and tensile pressure, increasing also the stress concentrations snd failure in these sections.

In order to reproduce the explosion and study the effect that the resulting pressure blast wave has on the structure, the Finite Element Method has to be implemented applying one of the three method explained in the following points:

- **The CONWEP method**: The name of this method comes from *Conventional Weapons Effects* and it consists on the direct loading of the target structure with the resulting pressure field produced by the blast through the introduction in the simulation software of temporal equations obtained with real experiments. Therefore, it is an empirical method. As advantage, it can be said that this method has a low computational cost.

- **The ALE or CFD method**: The Arbitrary Langrangian-Eulerian or Computational Fluid Dynamics method consists on the relation between the the pressure field that can generate the blast wave on the Langrangian structure and the fluid dynamics modeled with an Eulerian mesh. Hence, not only the target geometry has to be defined but also the fluid region, in this project the air that will make possible the propagation of the wave, in which the structure is going to be included. Applying this method, we can also obtained the effect of the waves reflexion when they reach the structure. This effect can produce an increment in stress concentration that has to be taken into account in the analysis. As a disadvantage, the computational cost is greater than the previous method.

- **The CONWEP-CFD method**: Both method explained before are combined in order to introduce pressures in the numerical model. In this case, the eulerian region, as it has been said before the air will be the fluid applied, is going to be reduced only to the structure environment, coupling between them. Through this coupling, the pressure field is transmitted from the fluid to the structure. Compared with the two previous cases, this method has the highest computational cost.

After analyze the three methods explained before, the CONWEP method will be finaly applied in order to develop this final project degree. [8].

2.4. *Abaqus* software

As it has been commented previously, before the technological advances that make possible accomplish projects as the one that is going to be developed instead of make experiments with the end of obtained a reliable database and find security solutions for air transport.

Once all the properties, characteristics and the suitable method is selected to solve the numerical problem, we need to choose the mean with which we will obtain the required results. This mean is a simulation software or CAE (Computer-Aided Engineering) software. For this project the simulation software selected is *Abaqus* included in the firm *Dassault Systèmes*.

This software use, as a resolution method, the FEM can that solve any kind of Mechanics Continuous Media problem in the engineering sector. *Abaqus* is divided into several modules, from the design one where all componentes can be design, if there is no a CAD (Computer-Aided Design) file to import in *Abaqus*, the addition of the material properties to each component, assembly of components if it is needed, interactions such as contacts or de definition of the blast as it will be explained in the following chapter, boundary conditions, mesh and time of simulation, and, finally, the visualization module.

Thanks to this software, different types of analysis can be perfomed such as static studies or dynamics with low velocities in which the mai objective is to obtain results with a high stress concentrations accuracy in structures. Static problems will be included in the module *Abaqus/Statics*.

The FEM can be also used to analyze a load effect in a period of time. This type of analysis are called non-linear analysis and they can be performed in the software *Abaqus* with the module *Abaqus/Explicit*.

After decide the topic which want to develop in the project, the next step in the process for obtaining the results of the problem is the selection of the most appropiate analysis between the two that have been explained previously. For this project, it has beed decided to implement the *Abaqus/Explicit* module as we want to study the effect of the blast in a delimited period of time. [9]

3. NUMERICAL MODEL

3.1. Introduction

In this chaper, every module in which Abaqus is divided will be explained not only in a general way but with the specific data that have to be included in order to achieve the goal of the project. The chapter will be divided in: Development of the model and the Failure Criteria. Starting from the creation of each components of the fuselage to the vizualization of the results.

3.2. Development of the model

In the picture (3.1), a passengers cabin from a real aircraft is shown.

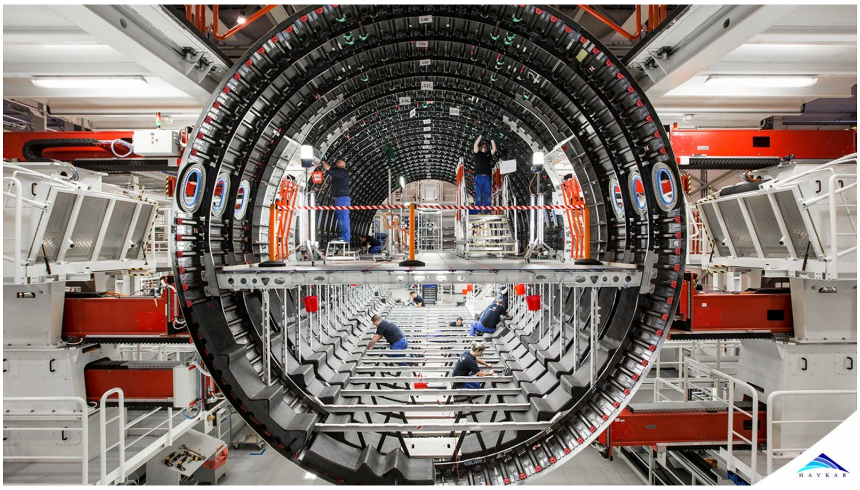


Fig. 3.1. Real passengers cabin, Photography credit: Havkar

Observing the figure (3.1), the components that will be implemented in the numerical model due to its importance, are shortly explained in the following points:

- **Skin**: 'The skin of an aircraft is the outer surface which covers much of its wings and fuselage'. (Wikipedia). [10]

Fig. 3.2. Fuselage skin. Photography credit: OGGturbojets

- **Frames**: Also called 'Former', it is a structure included in the reinforce fuselage system and it is used to give the cylindrical shape to the aircraft. (**Michael 1988**).

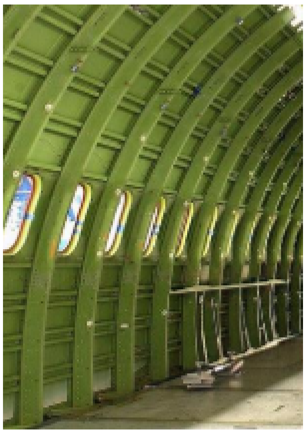


Fig. 3.3. Set of frames in a fuselage. Photography credit: Boeing

- **Vertical Stiffener**: Vertical component used to reinforce the structure in the vertical direction. They are located in the longitudinal direction of the fuselage.

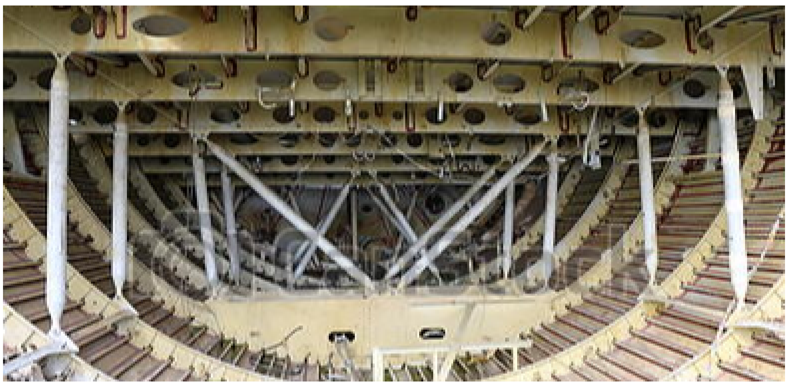


Fig. 3.4. Set of vertical stiffeners. Photography credit: CanStockPhoto

- **Floor**: It is the surface in which passengers stay during the flight. In this project two floor are going to be included in the fuselage.

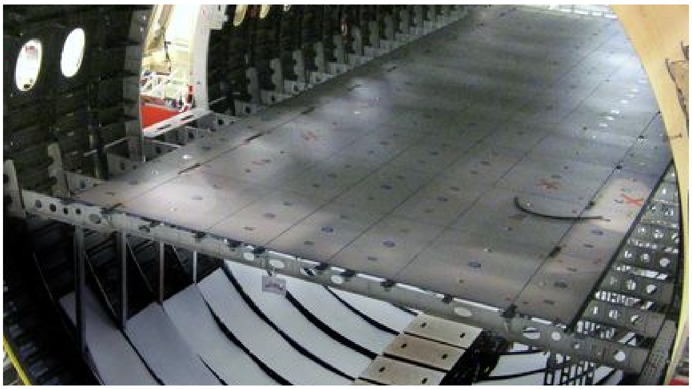


Fig. 3.5. Floor of the passengers cabin. Photography credit: Avitation

These five components introduced are going to be numerical modeled and simulated in *Abaqus*, following the process that will be explained in the *Abaqus* Modules Section.

3.2.1. *Abaqus* Modules

Part Module

The ojetive of the project is to analyze the effect of a TNT blast, taking into account that when the explosion reaches the structure, stresses are produced in it and, if they are greater than the Yield stress limit, the material goes under plastic deformation and comes the failure. In order to avoid an increment in stress concentrations due to details, such as holes for bolts, the model implemented in the numerical model is the simplest one. The dimensioning of the fuselage will be based on the A350 dimensions. [11].

Depending on the nature of each fuselage component, there will be two types of operations that are going to be used:

- ***Solid* operation**: 'A solid model is the standard solid element where the material is represented throughout the component/structure'. (Kuusisto, Eric. 2017). [12]. This type of operation will be used in non-sheet/plate components such as the frames or the vertical stiffeners.

- ***Shell* operation**: with a *Shell* operation the 'skin' of the model is modeled and it will be hollowed inside. 'Shells are a mathematical simplification of solids of special shape'. (Kuusisto, Eric. 2017).[12]. The skin and both floors are going to be modeled applying this operation.

Comparing both types of elements, it can be said that use *shell* elements means a huge decrease in the computational cost and, so, time saving, compared with the simulation of a *solid* element. [12].

The first component to be designed is the outer cylinder simulating the skin of the fuselage. This cylinder is designed with a *shell* type surface.

The fuselage has a length equal to 7.482 [m], a diameter of 5.971 [m]. As it has been designed with a *shell* surface type, a value for the tickness will be added in the **Property Module**.

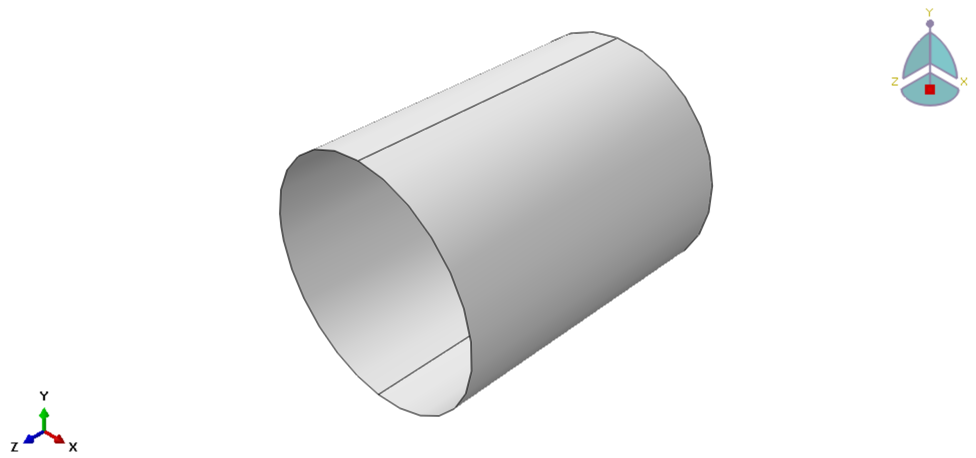


Fig. 3.6. Fuselage skin

Looking inside the fuselage, it can be seen two panels. The upper panel is modeled to simulate the floor where the passengers are going to be during the flight while the lower one is designed to divide the hold of the aircraft in two parts. The most important component of the fuselage and where the effect of the blast is going to be analyze is the upper panel, in order to preserve the integrity of this surface.

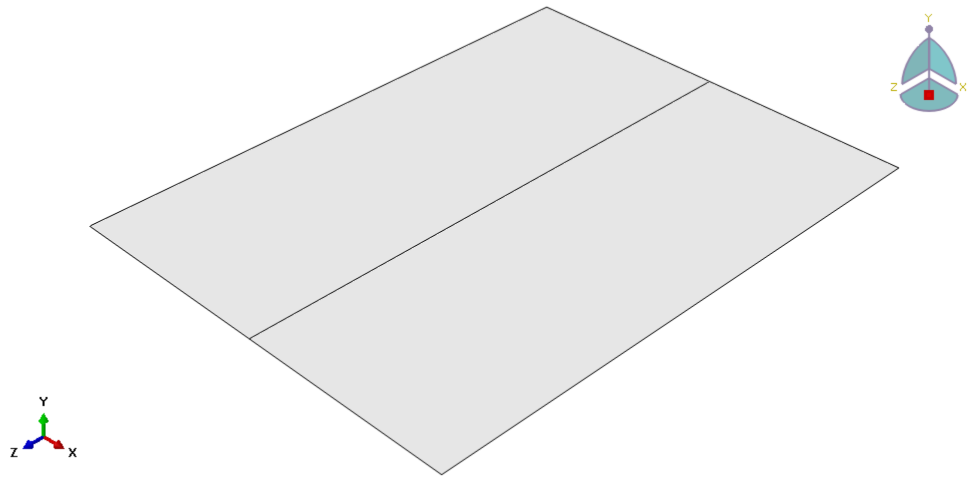


Fig. 3.7. Upper surface

Both panels are designed as a *shell* surface, the same procedure as in the skin. Delimiting the upper surface, it has a length of 7.482 [m] and width equal to 5.971 [m]. The width of this component is not exactly equal to the diameter of the skin due to the fact that to reinforce the skin, several frames are implemented in the fuselage, as it is going to be explained below. This surface is situated in the middle (vertical position) of the cylinder for passengers' comfort.

The position of the lower panel with respect to the upper one is 2.218 [m] below the upper floor. It is *shell* surface extrusion with 7.482 [m] long and it has a width equal to 3.7 [m].

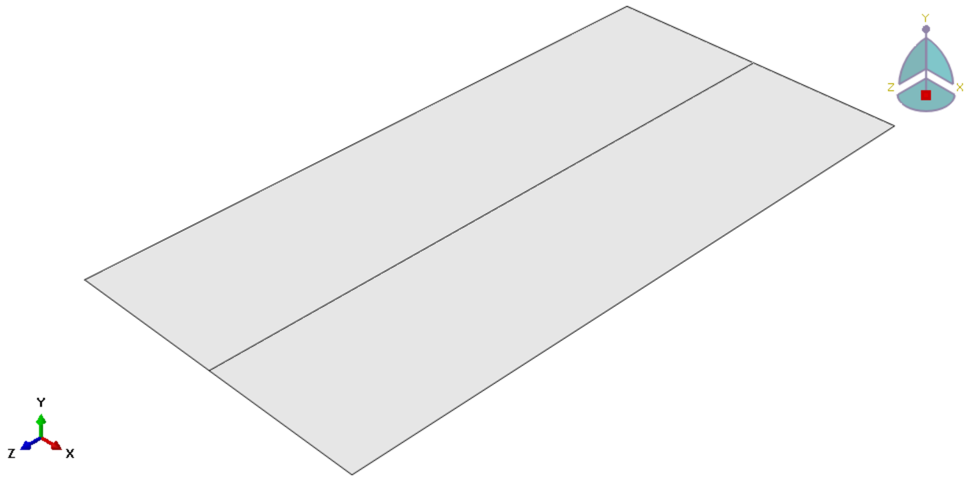


Fig. 3.8. Lower surface

Eleven frames have been designed to reinforce the skin and to connect it to both proposed panels to act like floors of the aircraft. These frames are separared from each other

0.7482 [m] to complete the length of the fuselage.

They are designed as a *solid* extrusion with a diameter equal to the diameter of the cylinder, that is 5.971 [m], and a width is 0.1 [m].

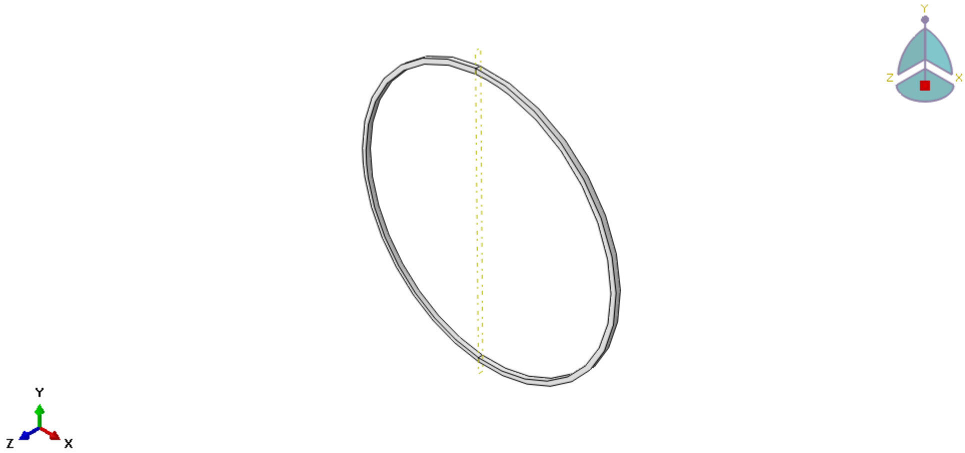


Fig. 3.9. Frame

The last component is the vertical stiffener. Their use is to connect both floors and reinforce the whole structure in the vertical direction. A matrix of 11 x 2 rows of stiffeners is implemented. They are a *solid* extrusion with 2.218 [m] long and a radius equal to 0.03 [m]. They are located where panels and frame are joined, where concentrations of stresses are going to increase due to the blast.

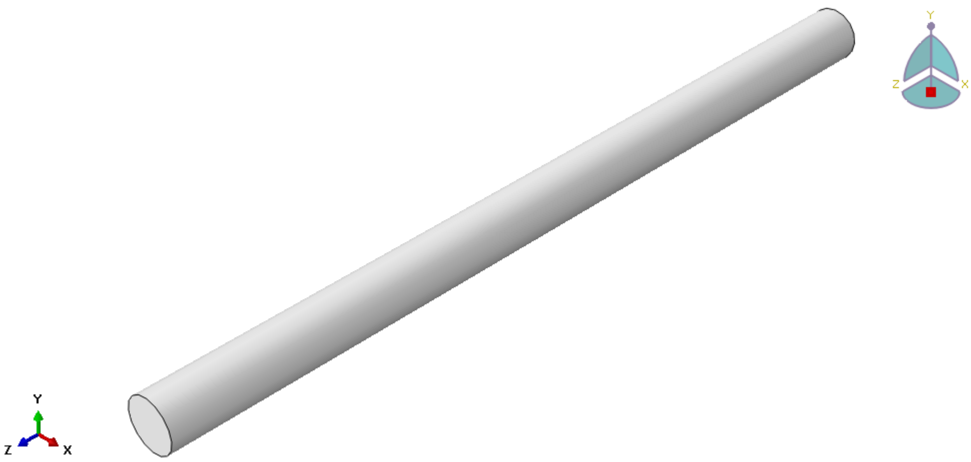


Fig. 3.10. Vertical stiffener

These last two components were designed with *solid* extrusion operation as they do not have a sheet geometry.

To sum up, the fuselage is formed by the following components:

Component	Type
Skin	Shell Extrusion
Frames	Solid Extrusion
Stiffeners	Solid Extrusion
Upper Floor	Shell Extrusion
Lower Floor	Shell Extrusion

Table 3.1. COMPONENTS OF THE AIRCRAFT

Property Module

In this section, the material selected is going to be configured. The elastic properties and the density of the Aluminum 2024-T3:

	Value	Units
Density	$2.768x10^3$	Kg/m^3
Young's Modulus	$7.3084x10^{10}$	Pa
Poisson's Ratio	0.33	

Table 3.2. DENSITY AND ELASTIC PROPERTIES

A material plastic behavior is defined through a set of constitutive equations, which show how the structure behaves under large deformations, high deformation velocities and high temperatures. ([13]).

To develop this project, the plastic behavior of the Aluminum is going to be defined with the Johnson-Cook Law due to the large deformation that the type of charge applied, a blast, can caused in the structure.

The equation, with which this law is modeled, is:

$$\sigma(\varepsilon, \varepsilon', T) = (A + B \cdot e^N)\left[1 + C \ln\left(\frac{\varepsilon'}{\epsilon_0'}\right)\right]\left[1 + \left(\frac{T - T_0}{T_f - T_0}\right)^m\right] \tag{3.1}$$

where ε is the plastic deformation and, thefore, ε' is the plastic deformation velocity. ε_0' is the reference plastic deformation velocity. Regarding the temperatures: T_f is the melting temperature, T_0 is the ambient temperature and T is the characteristic temperature of the problem.

To finish with the definition of the Johnson-Cook equation, the constants A,B,C,m and n represent: 'A represents elastic limit at ambient temperature with low deformation velocity conditions, B and n represent the hardness of the material due to the deformation and, C and n represent the sensitivity of the material to the velocity deformation and temperature, respectively' (Ruíz-Castro Alcobendas, Rodrigo. 2009.)

In order to adapt the equation (3.1) to the analysis that is going to be develop, it can be said that according to the main objective of the project, the deformation term is the only needed.

Therefore, the new equation to implement in the software *Abaqus* is:

$$A + B \cdot e^{N} \tag{3.2}$$

Once the constitutive equation for the Aluminum is defined, it is necessary to define a failure criterion for the same material.

Based on the failure criterion proposed by Hancock and Mckenzie (1976) [14], Johnson-Cook introduced the effects that the deformation velocity and the temperature can cause in the structure and, so, the simulation results. [15]. This way, a more general failure criterion could be applied.

This criterion is based on experimental parameters depending on the material and temperature values, as it is shown in the equation (3.3):

$$\varepsilon_f = \left[D_1 + D_2 e^{\frac{D_3 \sigma_m}{\sigma}}\right]\left(1 + D_4 \ln\left(\frac{\varepsilon'}{\varepsilon'_0}\right)\right)\left(1 + D_5\left(\frac{T - T_0}{T_f - T_0}\right)\right) \tag{3.3}$$

As it has been commented previously, due to the fact that the main objective of this project is to achieve the plastic deformation of the target surface and, based on experiments, the equation (3.3) can be simplified to the equation (3.4) because in order to study the deformation of a structure, the value of the parameter D_1 is enough. [13]

$$\varepsilon_f = D_1 \tag{3.4}$$

In the table (3.3), all the parameters related with both Johnson-Cook Law and Johnson-Cook Failure criterion, that are going to be introduced in the software *Abaqus*, are shown:

	Value	Unit
A	$2.3769x10^8$	Pa
B	$5.0129x10^8$	Pa
N	0.258143	
d1 (Damage)	0.2	

Table 3.3. PLASTIC AND DAMAGE PROPERTIES

When all the Aluminum properties are implemented, two different sections are going to be created: one for *solid* components and the other for *shell* ones. In order to be able to simulate the model, the addition of a thickness value for the *shell* componentes is needed. This way, the skin and both floor are going to have a thickness equal to 5[mm].

After defining the material and create the sections, it is necessary to associate the correponding section to each component. To associate them, the *Section Assignment Manager* tool is applied. The skin and both floors will have the *shell* properties, including the thickness of the surface, and frames and vertical stiffeners the solid properties.

Assembly Module

To obtain the final fuselage after designing the components and associate them the material properties, it is necessary to join all the components in the **Assembly Module**. Taking into account that vertical stiffeners follow a linear matrix as well as the frames, it is easy to put together all the components from both floors to the skin of the fuselage. The final assembly is shown in the image (3.11):

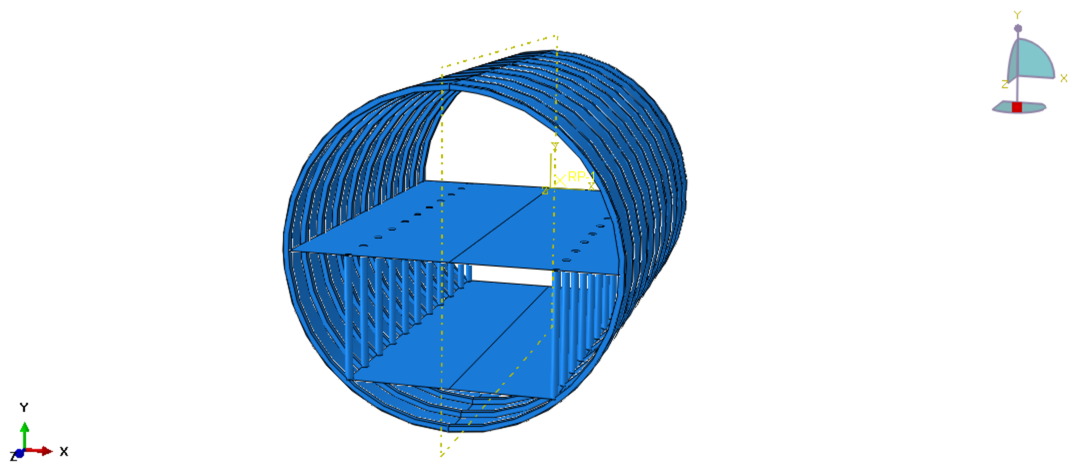


Fig. 3.11. Final assembly of the fuselage

The partition, that can be seen in the figure (3.11), has been made in order to reduce the time of simulation, appliying the symmetry that characterize the model. Creating a plane and using the *Partition* tool, the fuselage is divided into two halves. The following step is to mesh one of the halves, as the other half will have the same results.

Step Module

In the **Step Module**, the time that the explosion will last is introduced. After checking that the plastic deformation of the structure happens before 0.01 [s], as many studies from different sciencist reflects, the effect of time on the results has been also checked. Finally, it can be concluded that stresses do not increase by increasing the time of detonation, therefore, the value of 0.01 [s] will be implemented in *Abaqus*.

As a default, there is one step created but another step will be created to introduce the period of time for the blast, as well as, the type of simulation wanted, that is *Dynamic, Explicit*.

Based on what it has been introduced in the previous chapter, an *Explicit* analysis has been selected. This analysis selection can be explained due to problem dependency with time and the large deformations that can cause the type of charge introduced, a blast.

As the final results of the simulation depend on how the mesh is deforming as the time increases, if an *Implicit* analysis is applied, the mesh would have to be updated with each time increment, because the mesh starts to stop adapting correctly to the real deformation of the element and, therefore, results will not be as reliable as they are required.

Once the mesh is updated, the old mesh results will be now used as initial conditions for the analysis of the new adapted mesh.

In the other hand, if an *Explicit* analysis is implemented, the adaptation of the mesh to the evolution of the problem, is automatically done. Hence, the reseacher will only have to introduce the data correctly before starting the simulation. [16].

Interaction Module

This module is one of the most important ones because, as not only interactions between components are configured, but also the blast is defined in this section. Taking into account that components must be completely fixed bewteen each other, the *Constraint Tie interaction* tool is used.

A master and a slave surface should be defined in order to tie the upper floor to frames, lower floor to frames and frames to the skin and also between the upper floor and the vertical stiffeners. The result is shown in the figure (3.12):

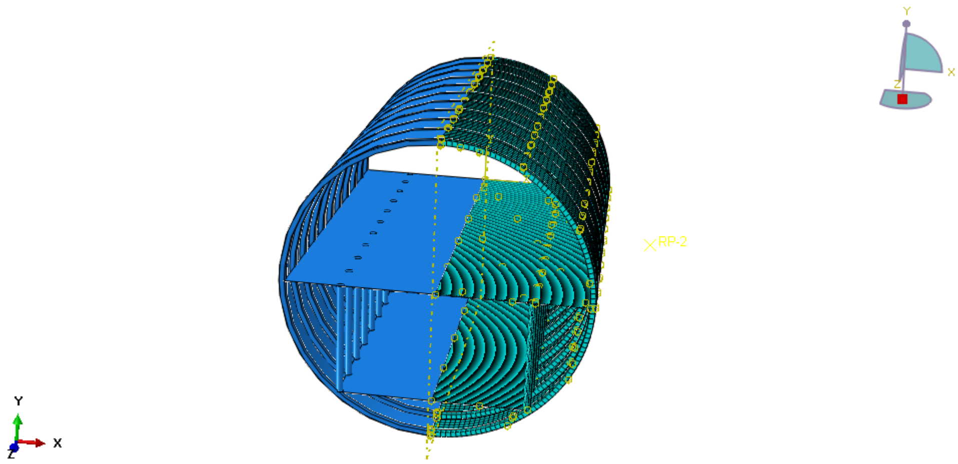


Fig. 3.12. Tie interaction between components

The definition process of the blast is made in the *Interaction Property Manager* and *Interaction Manager* .

First, the blast is defined in the *Interaction Property*. In this section, the mass of TNT to detonate and the conversion to [Kg] is introduced. Notice that it is an acustic air blast.

Secondly, in the *Interaction Manager* a reference point must be defined in order to associate it as the origin of the detonation. In this project, the reference point is going to be moved from the location simulating the cockpit to the one simulating the engine and, finally, to the hold of the aircraft. These three points have the following coordinates:

	X [m]	**Y [m]**	**Z [m]**
Cockpit	0	0	-1
Engine	3.986	0	3.741
Hold	1.493	1.109	3.741

Table 3.4. COORDINATES OF REFERENCE POINTS

In the same window, the upper surface is selected as the structure to be analyze, hence, only results regarding this component are going to be displayed in the Visualization Module.

After defining the reference points, with the *Interaction Manager* tool, the origin of the blast, and its properties, is associated to the reference point and the target surface where the effect of the explosion is going to be analyze. To model the blast, the CONWEP method, explained previously, will be used.

Loads and Boundary conditions Module

Starting with boundary conditions, due to symmetry only one half will have boundary conditions. This part will be restricted by the two ends of the fuselage by the skin and the upper surface, not only in displacements but also in rotations. This restriction will recreate connexions between the fuselage and the cockpit or the tail of the aircraft.

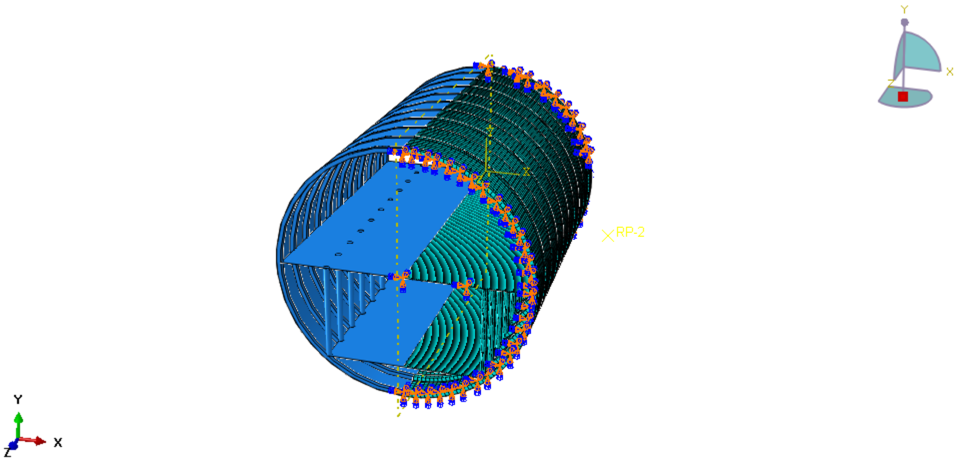


Fig. 3.13. Boundary condition

Mesh Module

Once all the components are designed with the material properties, they have to be meshed, meaning that components are divided into the same number of small finite elements to create the most realistic structure and obtain the most reliable results. The smaller the finite elements are the longer the simulation will take, therefore, a a smaller size on the target elements will be implement and increase the size of the elemetnts in components not important for the analysis. The geometry of the component is also considered as there are differences between create a mesh of a sheel and of a curved surface, as it has been explained before.

Only one half of the fuselage will be meshed, after applying the *Partition* tool, to reduce the time of simulation. The mesh of each component is shown:

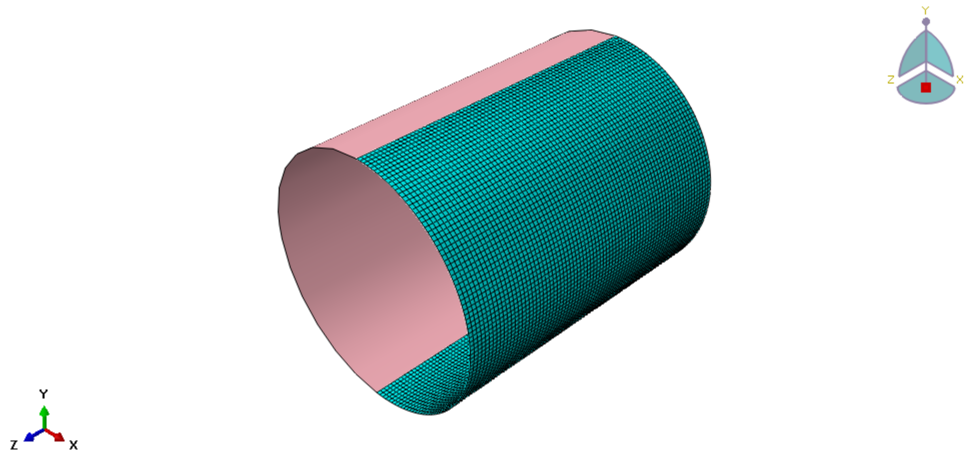


Fig. 3.14. Skin mesh

In this case, the curve of the skin has been taken into account to create the mesh of the skin. The half of the fuselage that will be simulated has been divided into 7050 (This value must be multiply by two in order to obtain the mesh of the whole skin) elements, with linear quadrilateral shape.

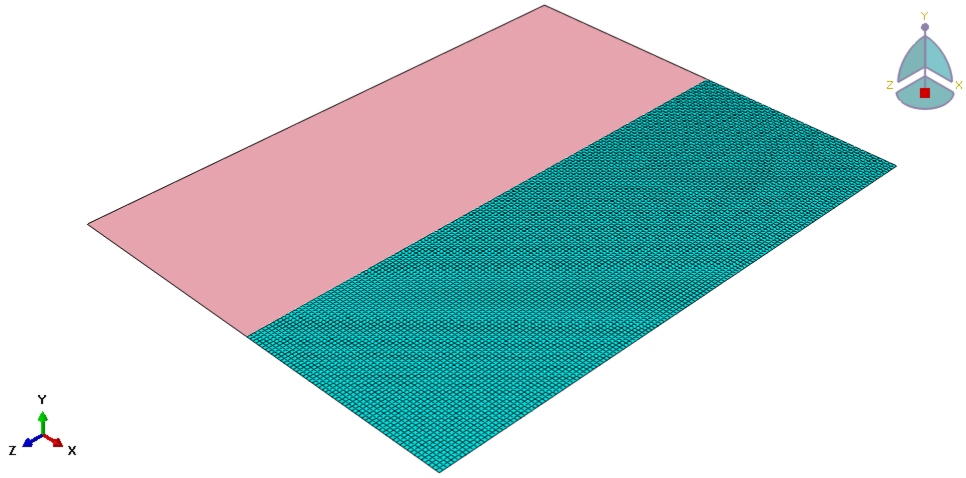


Fig. 3.15. Upper surface mesh

The mesh of the upper surface is shown in the figure (3.15). This is the target surface of the analysis and the half simulated has been divided into 8700 linear quadrilateral elements.

The lower surface used to divide the hold has been divided into 5550 also with a linear quadrilateral shape:

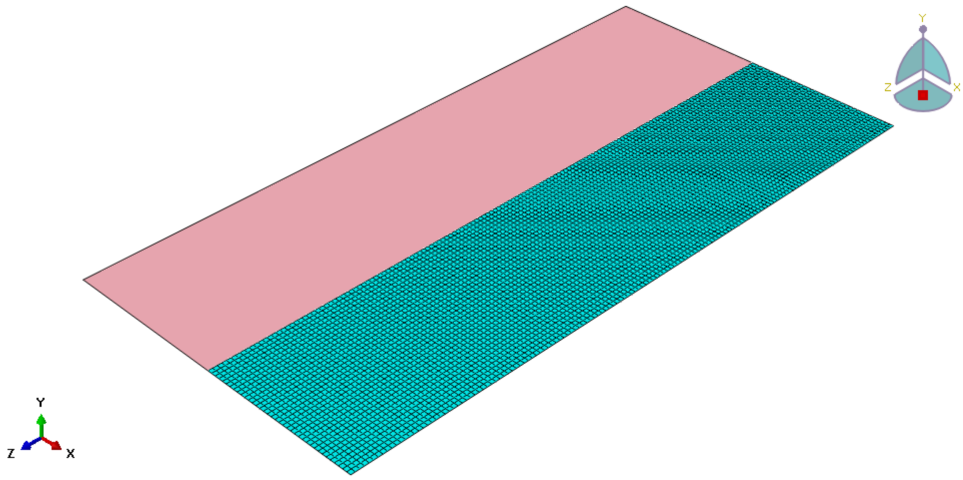


Fig. 3.16. Lower surface mesh

A half of the frame will be also mesh into 92 linear hexaedral elements as it is shown in the picture (3.17):

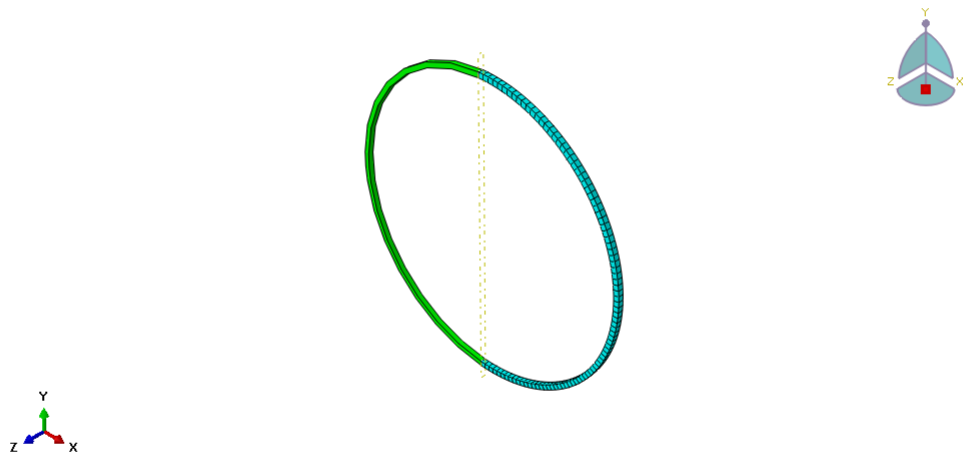


Fig. 3.17. Frame mesh

Concerning the mesh of the vertical stiffener, in this case it has no partition as the previous elements. It has divided into 78000 elements each vertical stiffener:

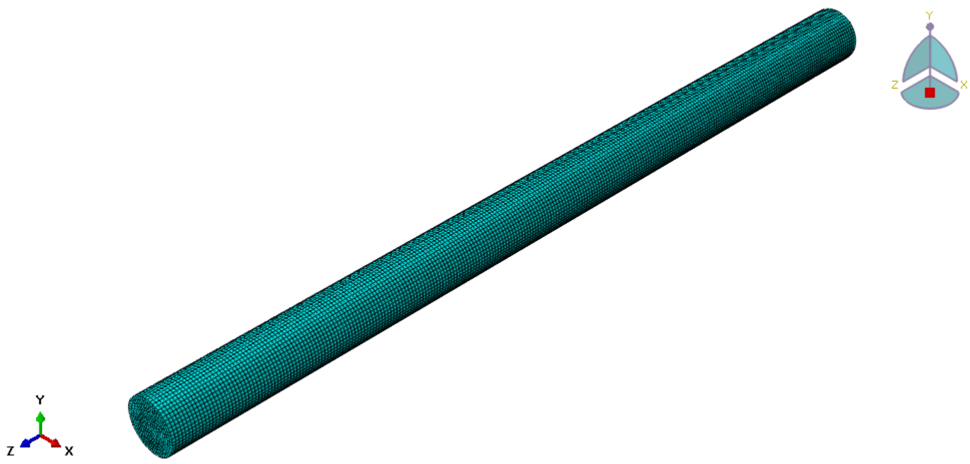


Fig. 3.18. Vertical stiffener mesh

Finally, in the figure (3.19), it is shown the mesh of the half fuselage that is going to be simulated. Notice that the final number of elements must be multiply by two in order to get the real value of elements in which the fuselage is divided. The number of elements is 880312 with linear hexaedral and linear quadrilateral shapes:

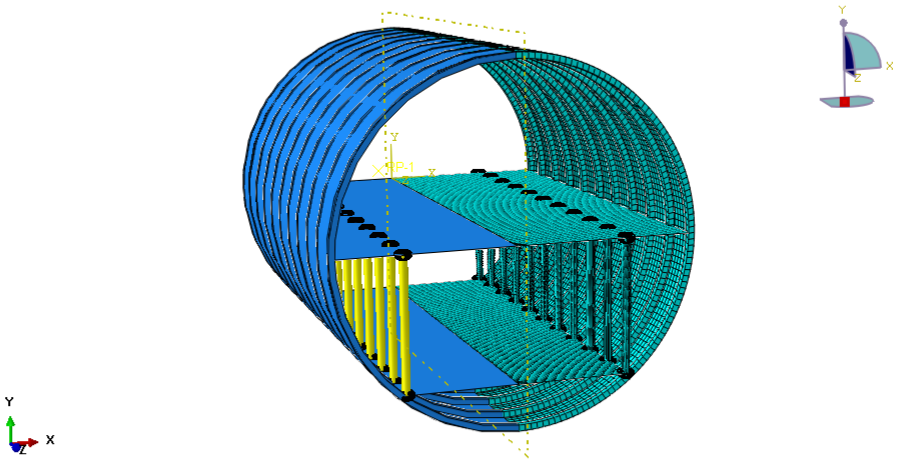


Fig. 3.19. Final fuselage mesh

This final mesh has been obtained after a sensibility analysis, therefore, the mesh displayed in the figure (3.19) is the optimal one, in terms of not only results but also in terms of the computational time.

Job and Module Visualization Module

After introducing all the properties and characteristics of the fuselage, a new job has to be created to start the simulation and the calculations. As soon as the simulations ends, all the results can be analyzed in the Visualization Module. To finish the process, displacements, stresses and pressure can be analyzed, once the blast wave reaches the target surface.

3.3. Failure Criteria

The objective of this Final Degree Project is to know at which mass of TNT will the aircraft fuselage proposed suffer the plastic deformation. When a load, a blast for this project, is applied on a structure, this one will start to suffer stress concentrations in the sections where boundaries conditions have been introduced or they may appear also where the different parts of the structure are joined together. This fact makes stress be the principal parameter to study.

Thanks to all the knowledge acquired during the degree about Elasticity and Strength of Structures as well as Aerospace Structures, the life of a component under a load depends on the value for the material Yield Stress applied to the component. 'This Yield Stress is the maximum value for the stress that an elastoplastic material can reach before going under permanent deformation'.(Wikipedia). At this point, the component starts to fail until the fracture, if the load continue.

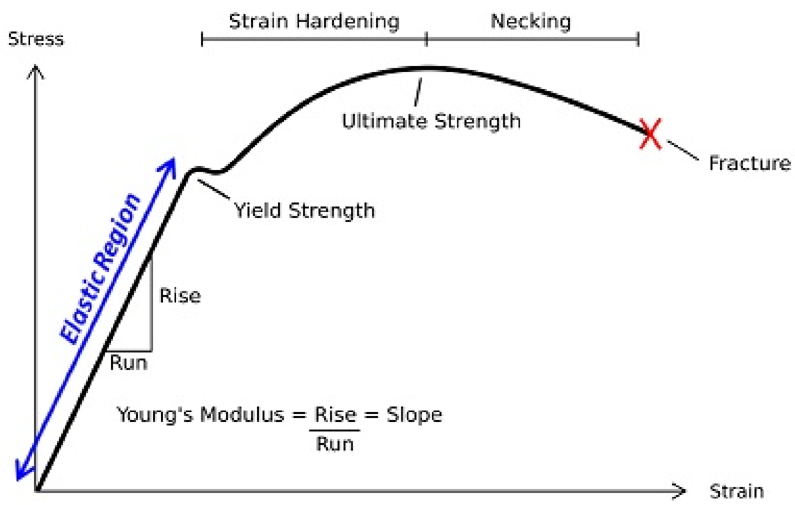


Fig. 3.20. Stress-Strain Curve. Photography credit: Instructables

To analyze this behavior, the Von Mises maximum stress criterion will be apply. Based on this criterion, when the Von Mises stress caused by a load is equal or greater than the material Yield stress, starts the failure of the component:

$$\sigma_{VM} > \sigma_{Yield} \tag{3.5}$$

4. ANALYSIS AND RESULTS

4.1. Introduction

As it has been explained before, in order to study the effect of the blast on the structure proposed, the FEM will be applied. With this method, the following parameters are going to be obtained: stress, displacement and pressure, through the unit displacement between nodes caused by the load. The symmetry of the structure has been taken into account to reduced the time of simulation and, therefore, the results obtained will be exactly the same for both halves.

Regarding the time of detonation, many values have been introduced in order to analyze if the time is a critical parameter for the plastic deformation of the structure. The period of time introduced was between [0.01-0.05]. Finally, it was concluded, that the effect of increasing this time is that the wave expands above the whole surface, changing the values of the stresses as it travels along it, but it does not increase their value.

Hence, the time of detonation that will introduce in every simulation proposed so as to obtain the mass with which the structure plastifies is 0.01 [s]. It will be checked that the plastic deformation is achieved at 0.01 [s].

In terms of the mass, the first simulation will have 1[Kg] mass of TNT and it will be increased until the plastic deformation of the target surface. The plastic deformation of the sturface does not necessarily have to occur in all the three configurations at the same time, therefore, the simulation process will be stopped when, in one of the three configurations, the stress reaches the Yield stress value.

Notice that the rest of the structure will remain undeformed, due to the fact, that the only target area consider for the analysis is the upper floor.

In the figure (4.1), the location of the three detonation points are shown in an schematic way:

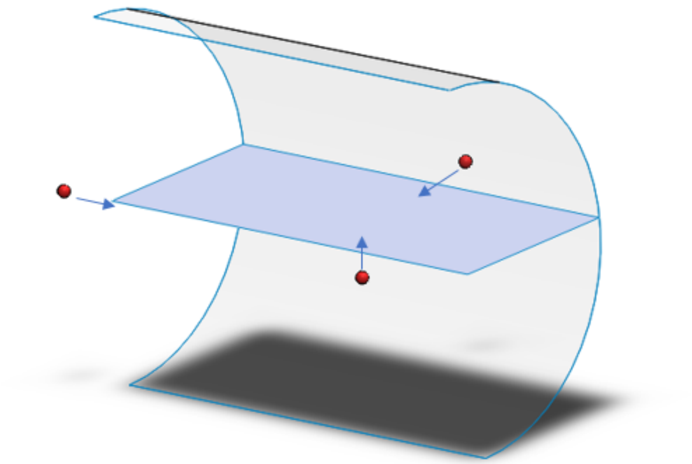


Fig. 4.1. Location of the detonation points related to the target surface

4.2. Simulations

Although many simulations have been made in order to achieve the main objective of the project, only two of them are going to be explained to show how the study has been developed. The set of simulations selected are: 1 [Kg] and 4.5 [Kg], that is the first and the last set of simulations.

4.2.1. Detonation of 1 [Kg] of TNT

First Simulation: In the cockpit

As it has been said in the introduction of the section, the problem analysis of the blast effect starts with the introduction of 1 [Kg] of TNT inside the cockpit. The first parameter to study is the displacement produced by the blast wave, while travelling along the target surface. As the most significant deformation of the structrure is in the vertical direction, the picture (4.2) shows it taking into account that the negative sign means that the movement is produced downwards.

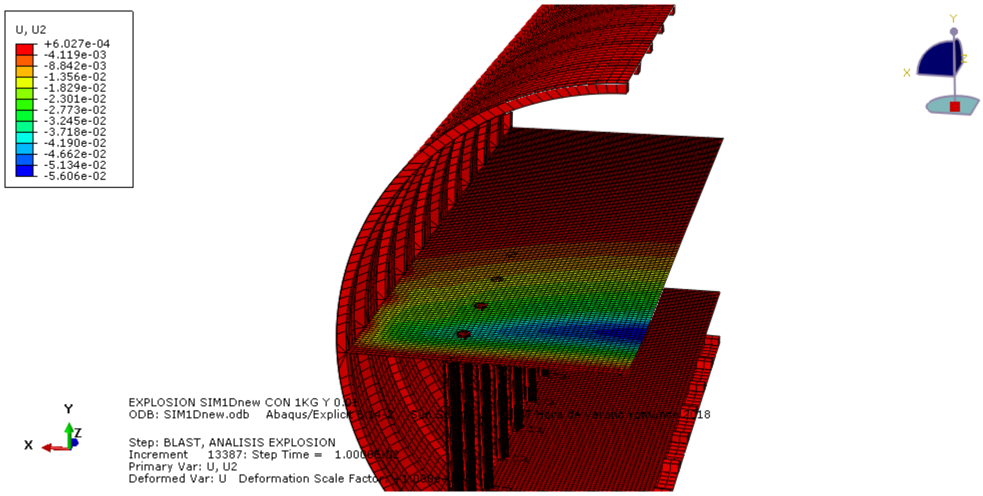


Fig. 4.2. Vertical displacements for 1 Kg of TNT in the Cockpit configuration

Considering that the blast origin is in the cockpit, the greatest displacements are produced in the surrounding of this focal point. It can be clearly seen how the waves moves along the surface and causing the deformation of the sheet. The highest value reach in a time of 0.01 [s] is $-5.606x10^{-2}$ [m].

Not only the surroundings are going to be deformed but the whole structure experiments displacements. The smallest one has a value of $6.027x10^{-4}$ [m] in the positive direction of the Y-axis.

The second parameter to analyze is the stress field:

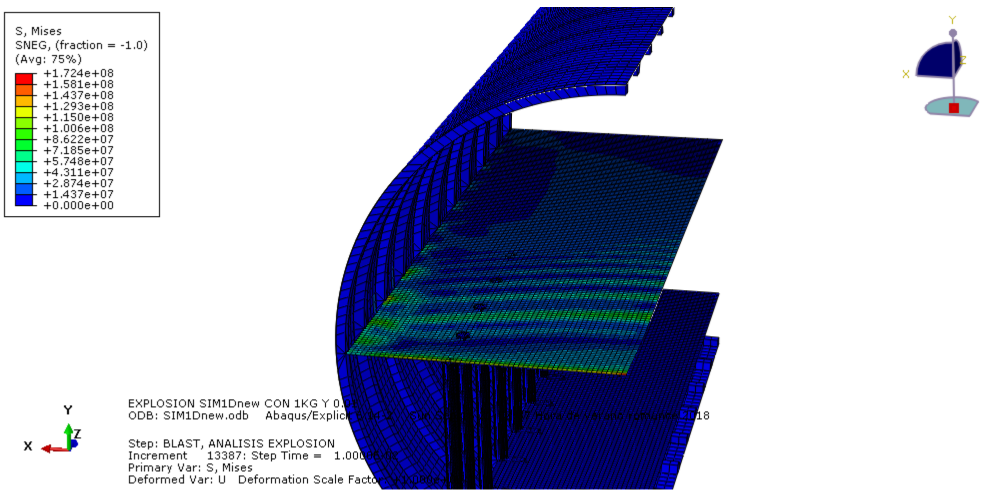


Fig. 4.3. Stress concentration for 1 Kg of TNT in the Cockpit configuration

It can be clearly observed how the wave affects the target structure, increasing the stress concentration where boundary conditions have been set. As it has been explained before, due to the boundary condition implemented, displacements and rotations are restricted in both ends of the upper surface.

These stress concentrations have a value equal to $1.724x10^8$ [Pa]. This section is not the only one with stress concentrations but also the section including the points where the upper surface joins the frames produce an increment in stresses. This increment is not as critical as the one in the surface ends but it is important enough to take it into account for later reinforcement of the area.

The pressure field is the last parameter to study:

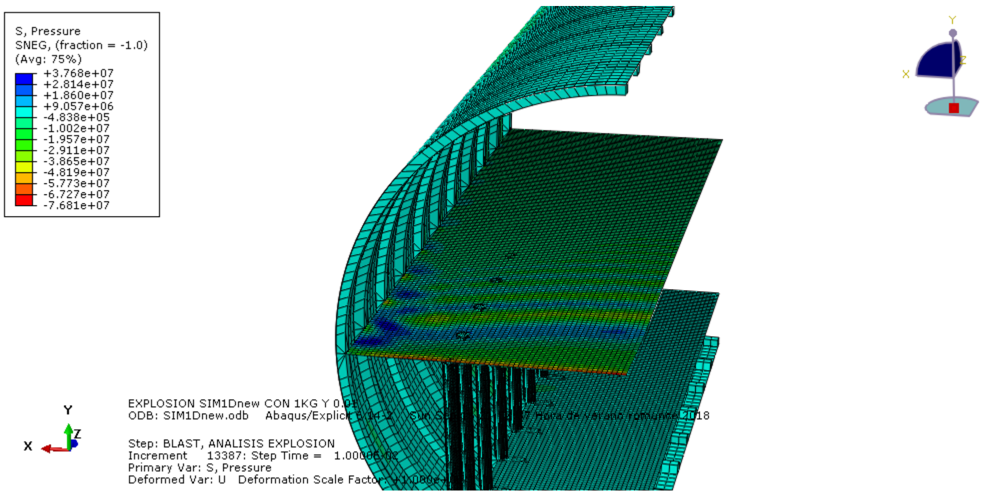


Fig. 4.4. Pressure field for 1 Kg of TNT in the Cockpit configuration

Comparing the results obtained for the stress and the ones obtained for the pressure, it can be observed that they keep a relation. Two types of pressure can be differentiated: the compressive pressure located in both ends of the upper surface, where boundary conditions are set. This compressive pressure concentration reaches a value equal to $-7.681x10^7$ [Pa] and is is represented by the red color shown in the color legend.

The other pressure shown is the tensile one. This pressure is located in the join section of the componentes, explained before. The tensile pressure, in this simulation, is greater than the compressive one and it has a value of $3.768x10^7$ [Pa]. In this case, the blue color represents tensile pressures.

Second Simulation: In an engine

For the second set of simulations, all the previous properties will remain the same, only changing the focal point of detonation from the cockpit to the lateral of the fuselage meshed, to reproduce an explosion in an engine.

Following the same procedure as before, the displacements in the structure are the first to be analyzed:

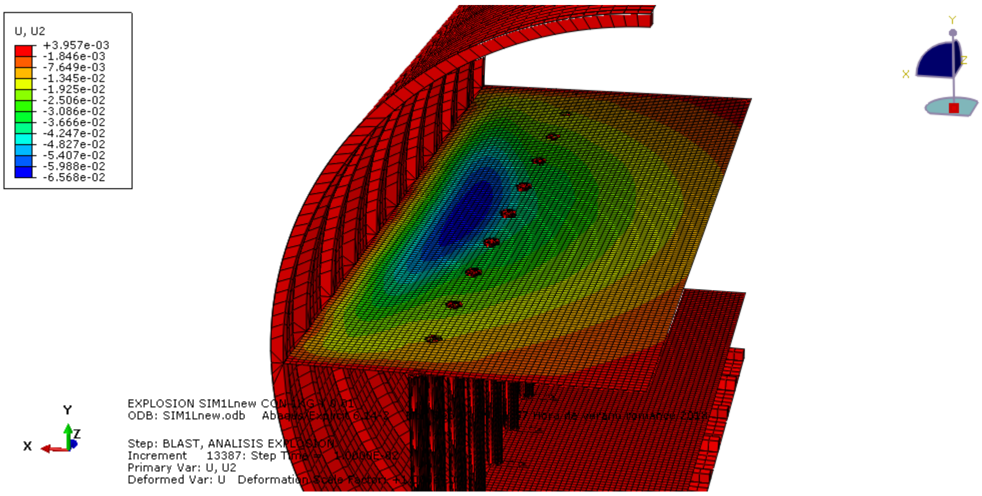


Fig. 4.5. Vertical displacement for 1 Kg of TNT in the Engine configuration

As it can be observed, the whole upper surface is more affected in this simulation than was in the cockpit one, although the direction of the deformation will remain the same, that is in the Y-axis negative direction with a value equal to $-6.568x10^{-2}$[m].

If the get away form the focal point, displacements decrease according to the color legend displayed in the picture. The smallest displacement experimented has a value of $3.957x10^{-3}$ [m].

The blast wave causes an increment in stresses in two main sections of the surface: at both ends of the target surface, where boundary conditions are set and where components are connected.

In the first case, as we do not have any movement of the structure, stress concentrations increases to a value of $1.173x10^{8}$ [m].

Regarding the second section named before, the highest values are not exactly in the point where components are connected but they are in the surroundings of the vertical stiffeners, as the wave has travelled to this point. These stresses are not as critical as the ones in the boundary conditions.

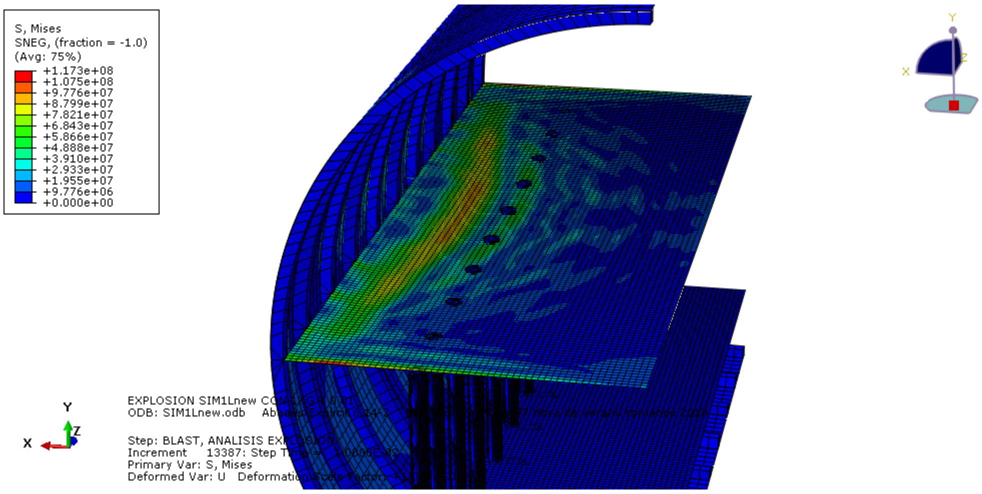


Fig. 4.6. Stress concentration for 1 Kg of TNT in the Engine configuration

The pressure field is divided into compressive and tensile pressure. The compressive ones are located in the boundary conditions as well as the stresses explained before. They have a value of $-5.819x10^7$ [Pa]. In this case, tensile pressure has a smaller value than the compressive one because boundary conditions restrict more than the interaction between components. Therefore, they apply a pressure of $3.768x10^7$ [Pa] on the upper floor.

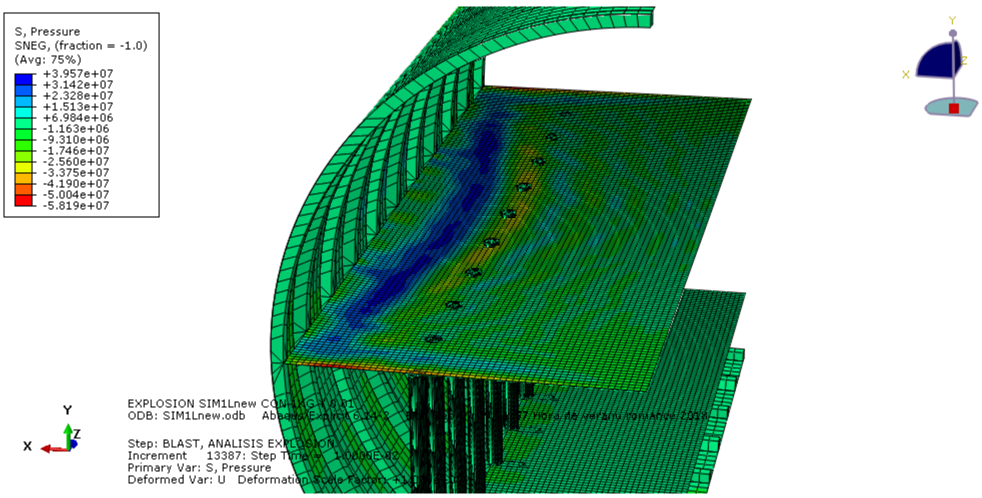


Fig. 4.7. Pressure field for 1 Kg of TNT in the Engine configuration

Third Simulation: In the hold

The final simulation of this group with, 1 [Kg] of TNT, is the one simulating the blast in the hold of the aircraft. Following the same procedure as in the previous two cases, the first parameter to analyze is the deformation of the structure due to the pressure blast:

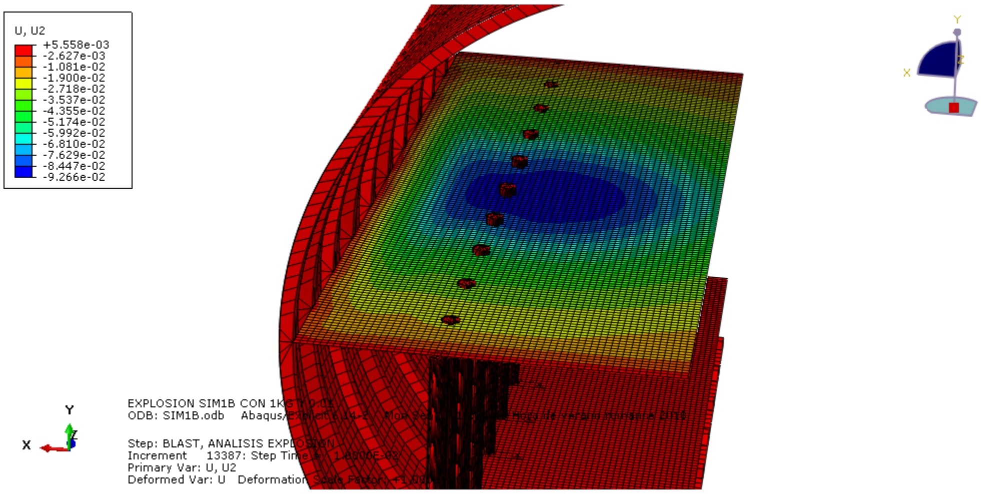


Fig. 4.8. Vertical displacement for 1 Kg of TNT in the Hold configuration

As the blast is coming from the center of the half hold, the greatest deformation of the floor has been produced in the middle section. According to the color legend, the blue color shows this displacement and it has a value of $-9.266x10^{-2}$ [m], that means downwards. The wave does not only affect this central section, but the rest of the surface is also affected, chaging from the value displayed before to $5.558x10^{-3}$ [m], upwards.

The unitary displacement of the nodes also causes stress concentrations located in the sections explained in the previous simulations. The stress field recorded by *Abaqus* is shown in (4.9):

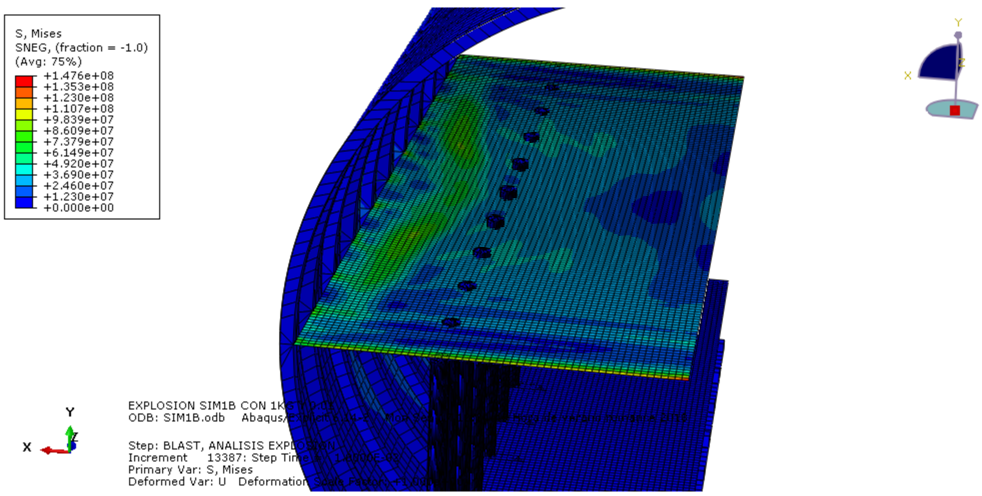


Fig. 4.9. Stress concentration for 1 Kg of TNT in the Hold configuration

Applying the Von Mises criterion, the plastic deformation of the structure is not obtained in the last simulation of this set as, the highest value recorded for the stress is $1.476x10^{8}$ [Pa], smaller than the Aluminum Yield stress.

This means that the mass of TNT should be increased to reach the main objective proposed for this project.

The last parameter is the pressure, compressive and tensile ones. The blast wave causes a tensile pressure near the frames and skin. This pressure has a maximum value equal to $3.404x10^7$ [Pa]. In the other hand, compressive pressures act near the boundary conditions. Values in this section are greater than the tensiles ones, that is $-6.453x10^7$ [Pa].

The effect of the pressure wave front can be observed in the image (4.10):

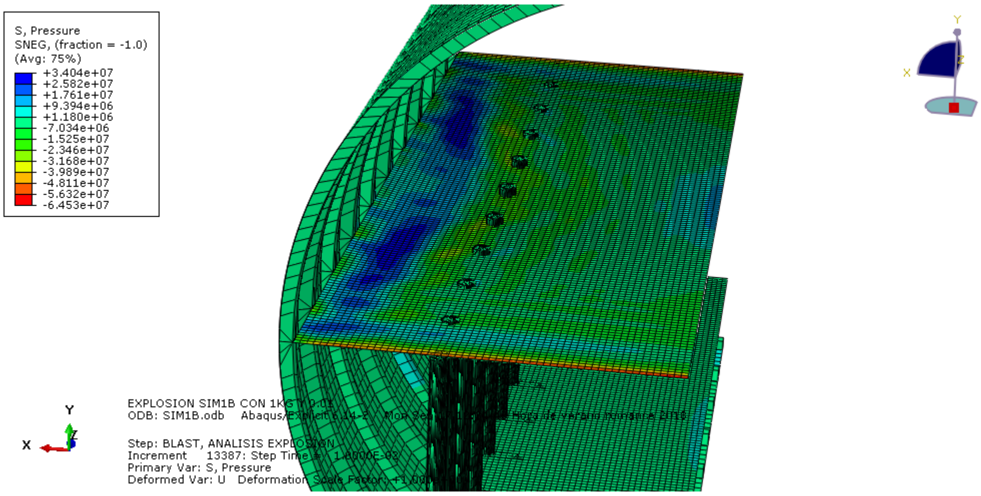


Fig. 4.10. Pressure field of the structure for 1 Kg of TNT in the hold

4.2.2. Detonation of 4.5 [Kg] of TNT

The final set of simulations, that are going to be explained will follow the same procedure of the previous ones, changing the mass to 4.5 [Kg] of TNT.

First Simulation: In the cockpit

As it has been done with the previous set of simulations, the vertical displacement results caused by the blast, with focal point of detonation in the cockpit, are displayed in the figure (4.11). As the blast wave does not reach the opposite edge of the upper surface, displacements are concentrated in the blast origin surroundings.

At a time of 0.01 [s], as it can be observed in the results, the most affected section is at the second vertical stiffener and, so, this section is the most deformed one due to the blast, reaching a value of $U = -1.237x10^{-1}$ [m], based on the color legend.

The negative sign in the value means that the displacement is in the negative direction of the Y-axis.

The rest of the surface is not very affected, obtained a final value for the vertical movement, in the Y-axis positive direction, equal to $U = 2x10^{-3}$ [m].

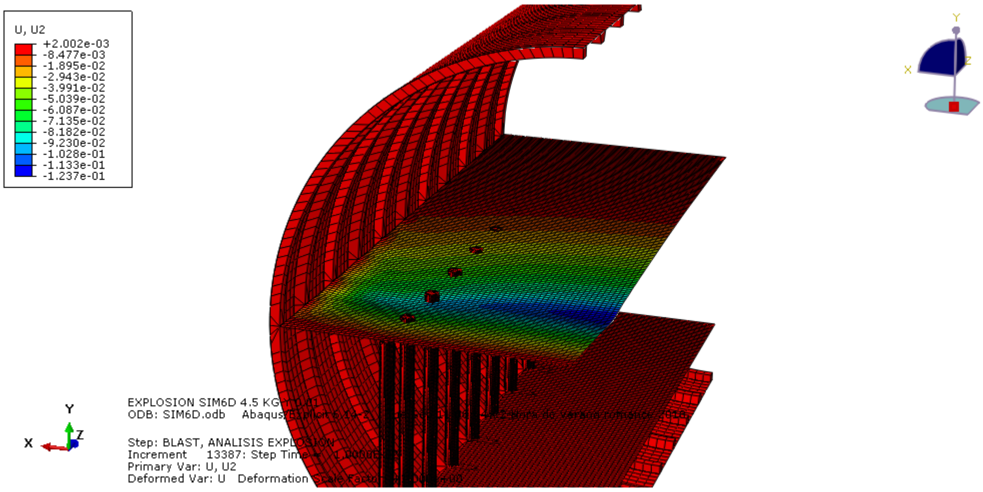


Fig. 4.11. Vertical displacement for 4.5 Kg of TNT in the Cockpit configuration

The stress results are displayed in the figure (4.12):

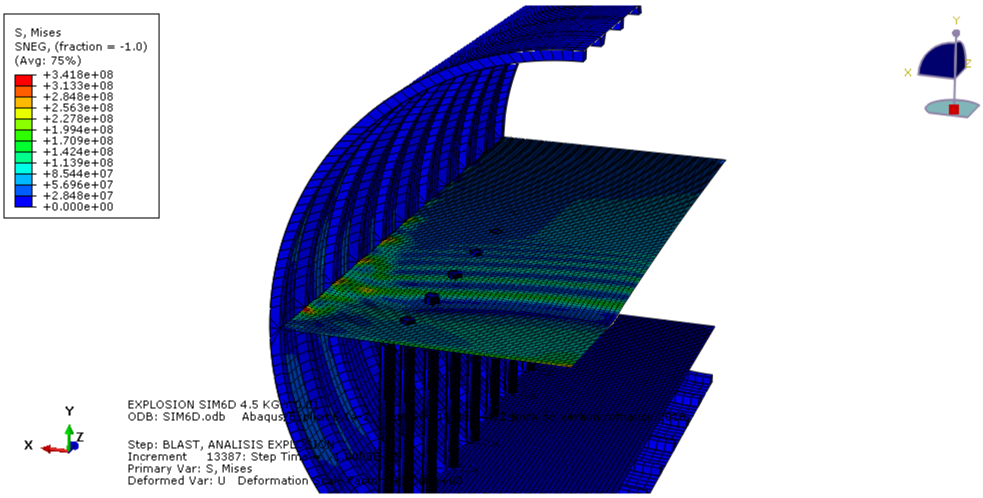


Fig. 4.12. Stress concentration for 4.5 Kg of TNT in the Cockpit configuration

In this case, the detonation of a mass equal to 4.5 [Kg] of TNT, with origin in the cockpit, produces an increment in stress concentration of the critical points located where components join. This increment reaches a value equal to $3.418x10^{8}$ [Pa], very closed to the Yield stress value of the Aluminum, although this value last a short period of time due to variable load applied as a travelling wave.

The last parameter to study is the pressure produced by the blast wave on the upper surface:

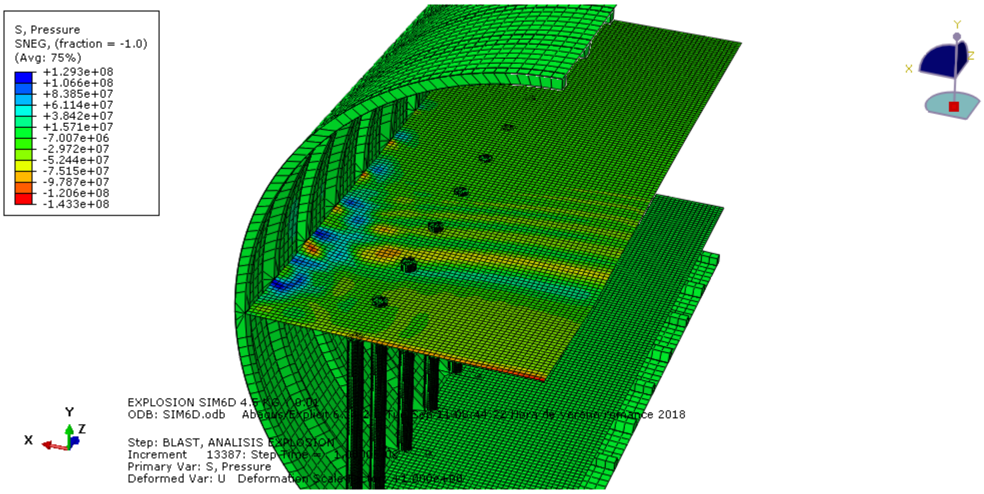


Fig. 4.13. Pressure field for a 4.5 Kg of TNT in the Cockpit configuration

There can be found both compressive and tensile pressures, following the color legend displayed in the picture (4.13). Where the upper surface joins the frames, both types of pressures affect the surface. Although the tensile ones are the predominants, the value of the compressive pressure is greated and equal to $1.433x10^8$ [Pa]. The highest value for the tensile pressure in this simulation is $1.293x10^8$ [Pa].

Notice that in the edges of the upper surface, where either displacements and rotations are restricted, there is also a compressive stress concentration with the same value as the in the other critical section.

Besides both stress concentration sections explained before, where the maximum displacement is achieved, stresses in other sections have increased too, but they do not reach the maximum values.

Finally, it can be said that both critical sections named before must be reinforced to avoid plastic deformation and failure of the sections.

Second Simulation: In an engine

In this second simulation, the origin of the blast has been changed to the left-hand-side of the fuselage, so as to reproduce the effect of explosion in an engine. The rest of the characteristics, regarding the mass of TNT and the material applied, are the same as in the Cockpit configuration.

In this simulation, the blast wave affects the whole upper surface, although the greatest displacements are produced where vertical stiffeners are situated, in the longitudinal direction. As in the previous cases, the negative sign, before the value of the displacement, means that is produced in the negative part of the Y-axis. It is equal to $-1.512x10^{-1}$ [m].

The movement of the rest of the structure is not as significant as the one explained before, but it reaches a value equal to $7.835x10^{-3}$ [m] in the Y-axis positive direction.

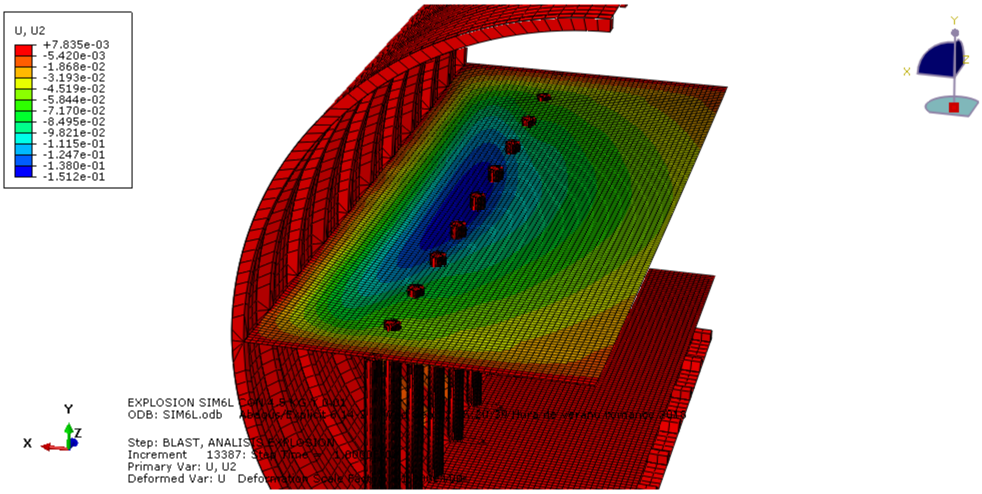


Fig. 4.14. Vertical displacement for 4.5 Kg of TNT in the Engine configuration

Talking about the stress field generated, stress concentrations are situated again where the upper surface and the frames join together. This section has a value equal to $3.043x10^{8}$ [Pa].

The upper surface edges are also affected by the blast wave, reaching the maximum value for the stress recorded.

Finally, in the vertical stiffeners surroundings there is a stress concentration but not with the critical value.

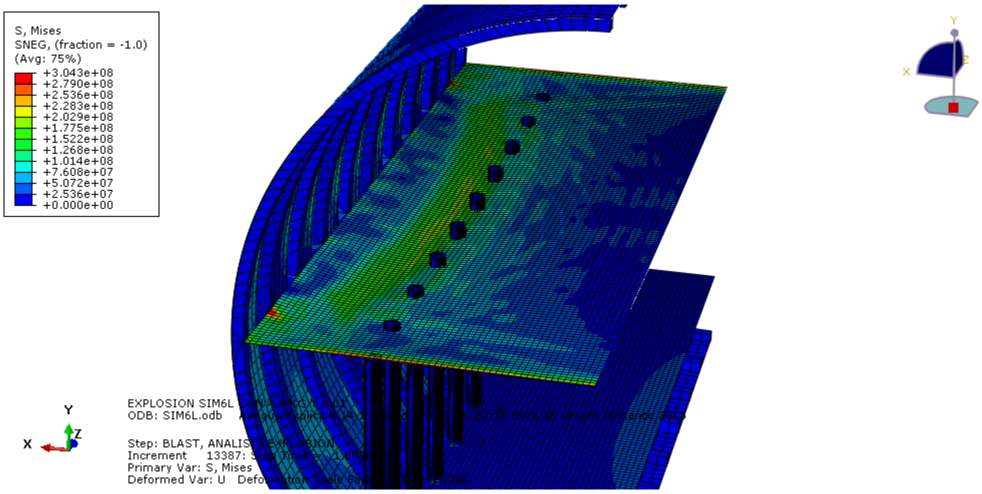


Fig. 4.15. Stress concentration for 4.5 Kg of TNT in the Engine configuration

The pressure field caused by the blast coming from an engine is shown in the figure (4.16):

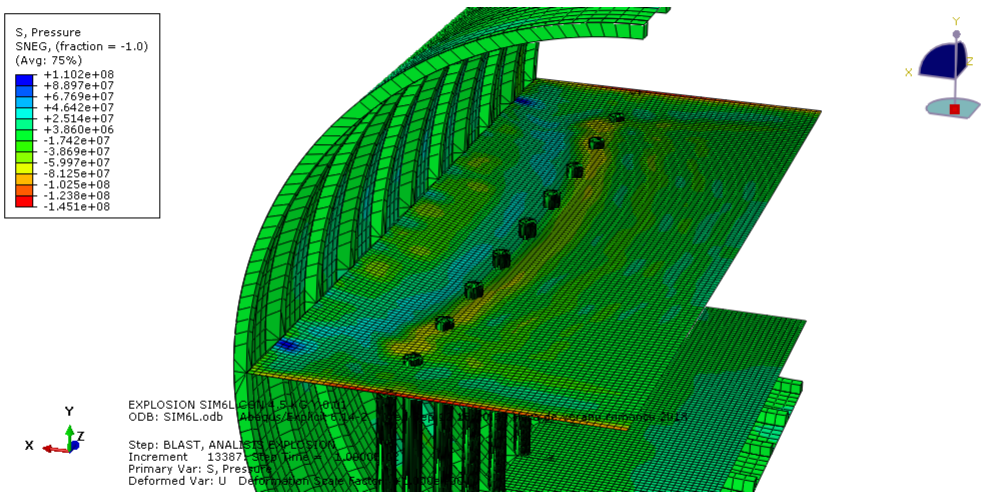


Fig. 4.16. Pressure field for 4.5 Kg of TNT in the Engine configuration

The stress and the pressure field are related, explaining why there is a concentration of tensile pressures where the passengers' cabin floor gathers the frames, reinforcing the skin. This tensile pressure concentration has a value equal to $1.102x10^8$ [Pa].

Addionally, there are compressive stress concentrations also in the edges of the upper surface, with a value equal to $-1.451x10^8$ [Pa]. In this case, the compressive pressures have a higher value compared to the tensile ones.

Third Simulation: In the hold

The last simulation with 4.5 [Kg] of TNT will be detonated in the hold of the aircraft. The rest of the simulation characteristics will not change.

Starting with the displacements produced by the blast, it can be observed in the following picture that the greatest deformations in the structure are located above the focal point of detonation. This deformation has a value equal to $-1.962x10^{-1}$ [m] in the Y-axis negative direction, that is downwards.

In this projects, as the only target surface is the upper floor, vertical stiffeners remain in the same position, as well as the skin.

The rest of the surface is deformed too, according to the color legend displayed in the results, with the smallest value for displacements on the fuselage equal $7.685x10^{-3}$ [m].

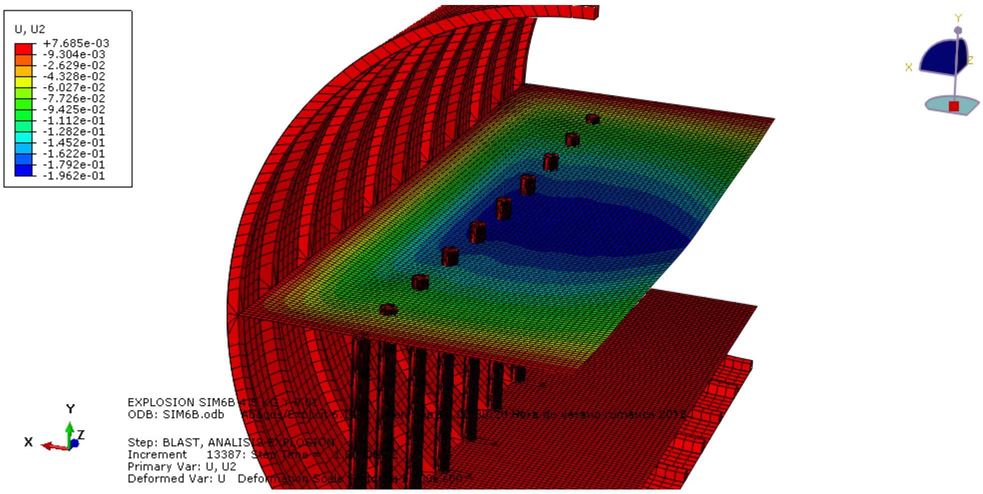


Fig. 4.17. Vertical displacement for 4.5 Kg of TNT in the Hold configuration

As the blast wave is travelling, not only the deformations on the structure will vary, but also the stress field. The stress results are shown in the figure (4.18):

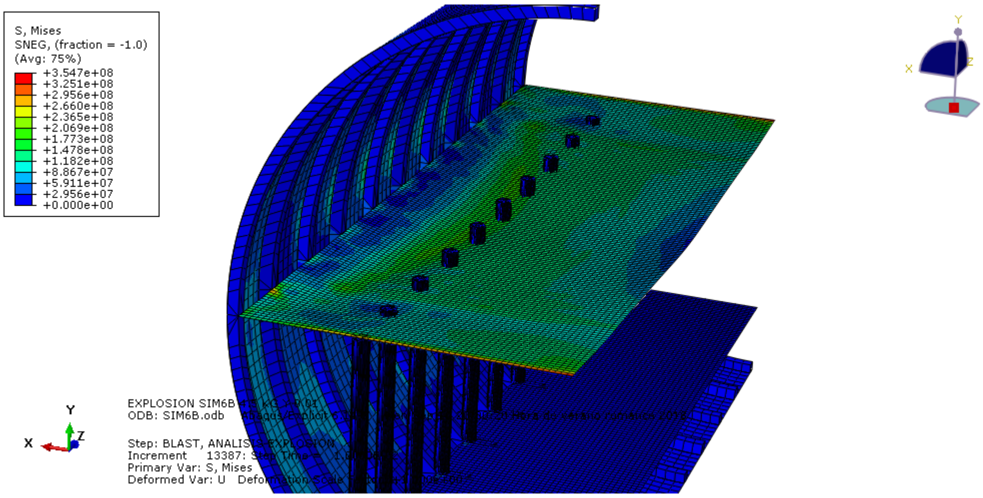


Fig. 4.18. Stress results for 4.5 Kg of TNT in the hold

Applying the Von Mises criterion, the Von Mises stress in this simulation is equal to $3.547x10^8$ [Pa]. Taking into account that the Aluminum 20241-T3 Yield stress is equal to $3.5163x10^8$ [Pa], means that the plastic deformation of the surface has occurred, according to (3.5).

This plastic deformation of the structure is located where the displacements and rotations are restricted, that is in the ends of target surface.

At this point, the structure will not recover the its initial shape, as this deformation is permanent. If there were more blast waves, according to the graph (3.20), the next step in the failure process of the upper floor will be reaching the Ultimate stress.

The Ultimate stress separates two phases of the structure in the plastic deformation region: before reaching the Ultimate value, the permanent deformation of the structure starts to be predominant due to the continued application of the load, that is the 'Strain Hardening' of the fuselage. Once the Ultimate stress is reached, the size of the section where the plastic deformation has taken place decrease. This fact is called 'Necking' and it leads to the fracture of the structure.

The last parameter included in the analysis of the structure is the pressure:

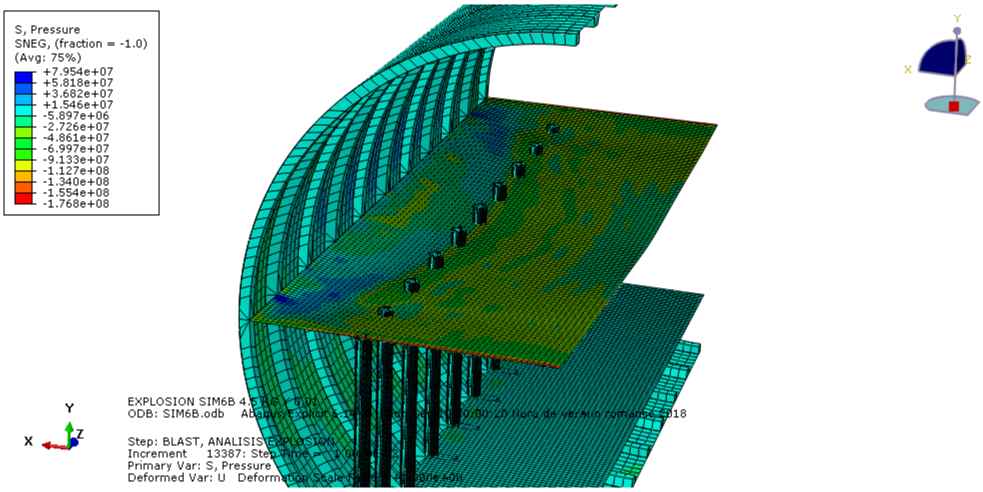


Fig. 4.19. Pressure field for 4.5 Kg of TNT in the Hold configuration

Considering the color legend shown in the results, the compressive pressure field is located where the plastic deformation of the structure takes place, that is in the upper floor boundary conditions. It has a maximum value equal to $-1.768x10^8$ [Pa] and it is colored in red.

The tensile pressure field is mostly located where the target surface joins with the frames, reinforcing the skin of the fuselage. In this simulation, the tensile pressure value is smaller than the compressive one and it is equal to $7.954x10^7$ [Pa].

4.2.3. Plastic Deformation in the Cockpit and Engine/Lateral Configurations

Although the plastic deformation of the structure is reached in the Hold configuration with 4.5 [Kg] and, therefore, the objective of this Final Degree Project, in this section the TNT mass with which the other configurations also reach the objective are shown.

The plastic deformation of the surface, in the Cockpit configuration, is not so far from the Hold one. Only 0.5 [Kg] of TNT was added in order to get a stress value equal to $3.516x10^8$ [Pa]. As the stress value is the base of the analysis of the results, it is shown in the figure (4.20) for this configuration:

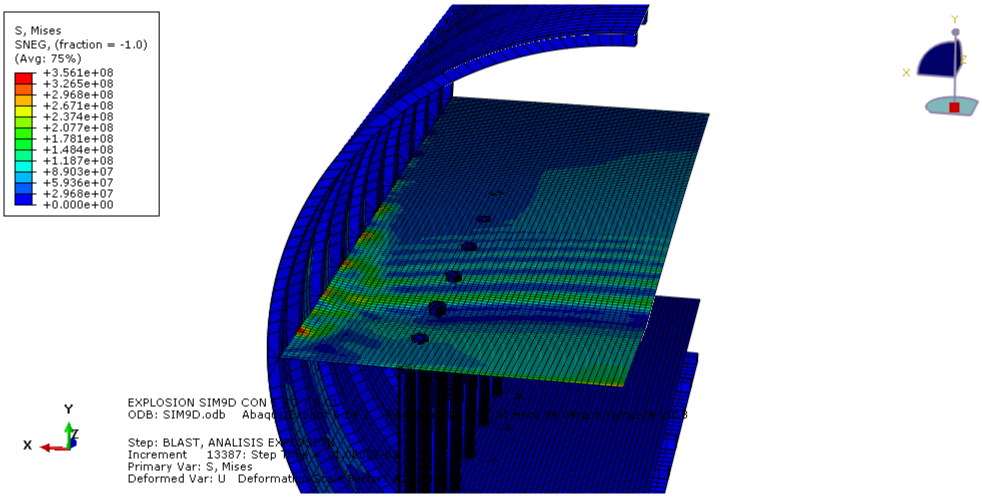


Fig. 4.20. Plastic deformation in the Cockpit configuration

In terms of the Engine or Lateral configuration, the plastic deformation is not reached until the TNT mass increases to 9 [Kg]. This fact makes this configuration the least critical, in terms of the preservation of the integrity of the upper surface. The same way as in the Cockpit configuration, the stress field caused, in the structure, is shown in the figure (4.21) with a maximum stress value of $3.647x10^8$ [Pa]:

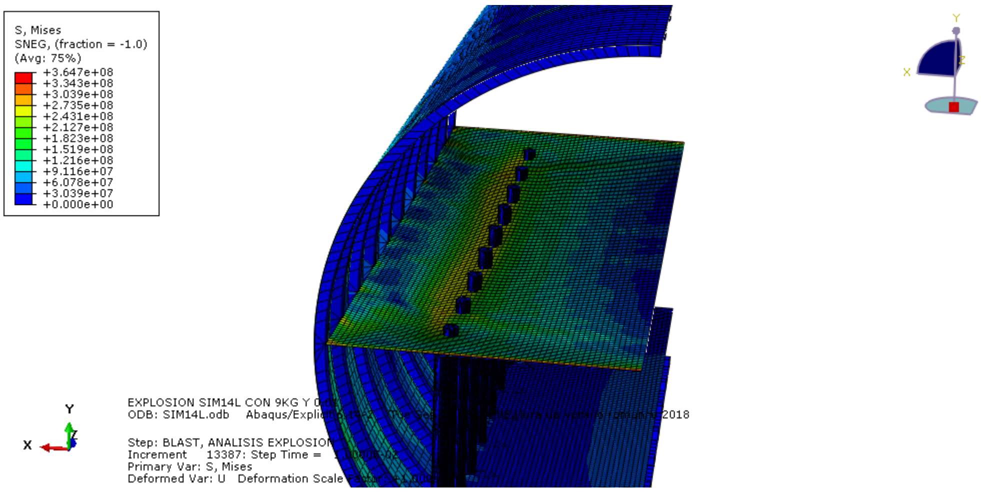


Fig. 4.21. Plastic deformation in the Engine/Lateral configuration

4.3. Evolution of the results

In this section, the evolution of the results, regarding the critical Hold configuration, are going to be commented due to fact that is in this simulation in which the plastic deformation of the upper surface has been reached.

The analysis will be focused on the stresses and displacements caused by the blast in the defined period of time [0-0.01] [s]. The section can be divided into the evolution of the results for the 4.5 [Kg] hold configuration with two plots and other two plots for the development of the results with the increment of the TNT mass, to achieve the objective.

Therefore, the first two plots are:

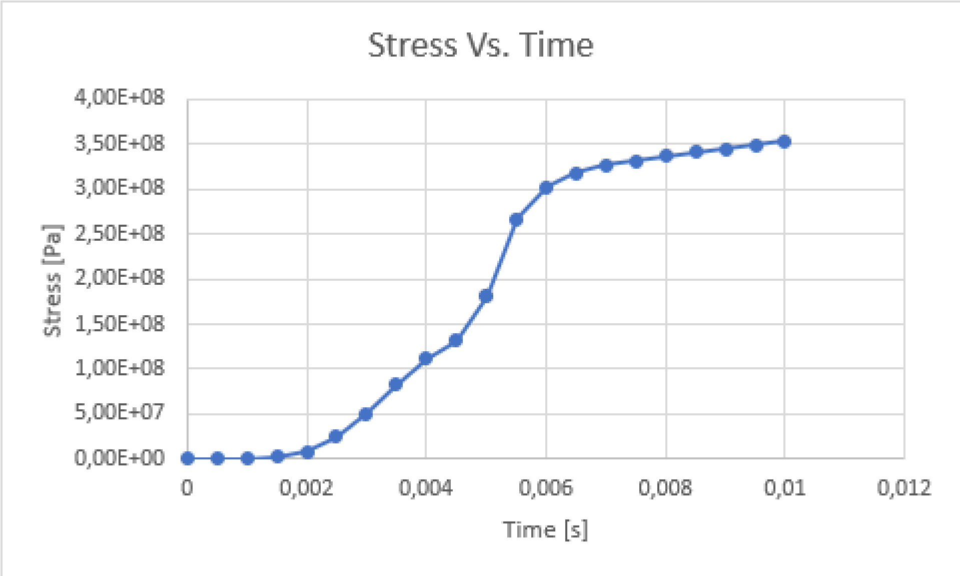


Fig. 4.22. Evolution of the stresses with time

It can be clearly seen the time at which the blast wave reaches the target surface and, from this point, its propagation along the surface. As the wave travels, stresses start to increase from t=0.002 [s] (touching point) to the end of the period introduced, t=0.01 [s], where the Von Mises stress overtakes the Yield stress.

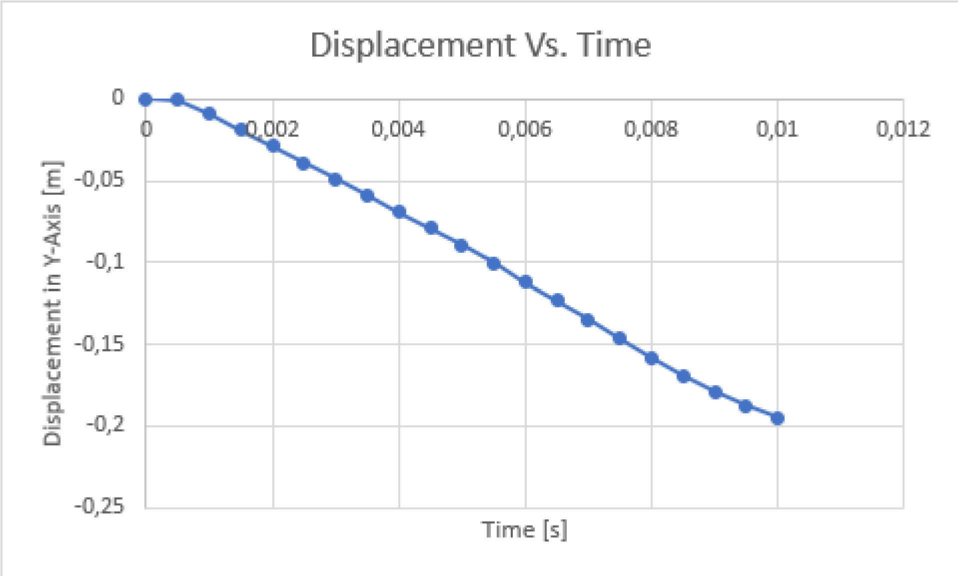


Fig. 4.23. Evolution of the displacement with time

In terms of the displacements, as it has been said previously, they are produced in the negative direction of the Y-axis. As the time increases, the movement increases in a linearly way, shown in (4.22).

The other part of the section is formed by two plots showing the evolutions of the results simulations with the increment in TNT mass, to find the plastic deformation of the structure, as said before. The evolution of stresses is the first plot to analyze. As the TNT mass has been increased, the stress recorded was also increasing.

The maximum value is the one for the critical simulation, that is the 4.5 [Kg] hold explosion, where the goal of the project is achieved.

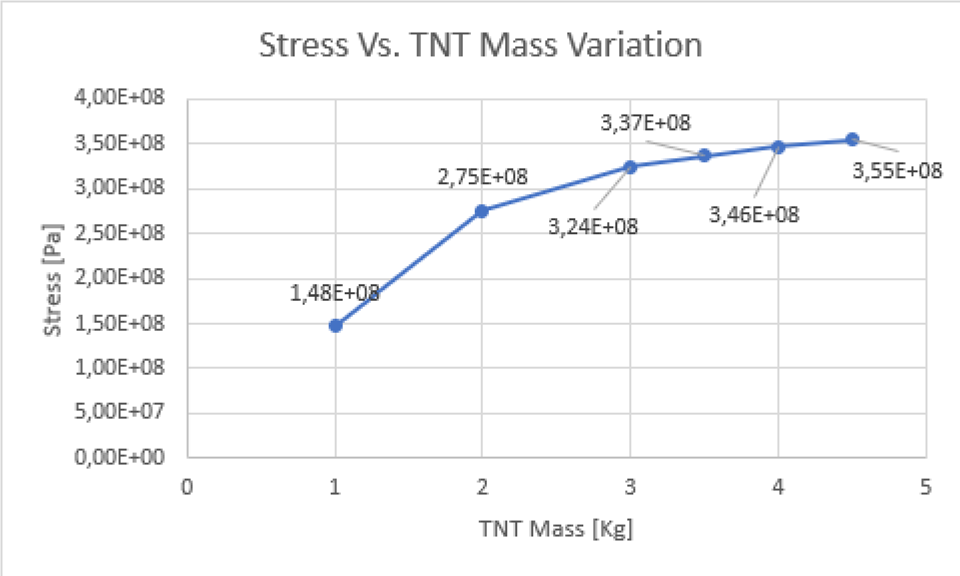


Fig. 4.24. Evolution of stresses with TNT mass

The last plot to comment is the evolution of the displacement with the TNT mass. Taking into account that the displacement values have been introduced with a negative sign due to its downward movement, it can be seen that also these results increase when a higher quantity of TNT is added to the explosion. The maximum value recorded for displacements is the one belonging to the 4.5 [Kg] hold configuration.

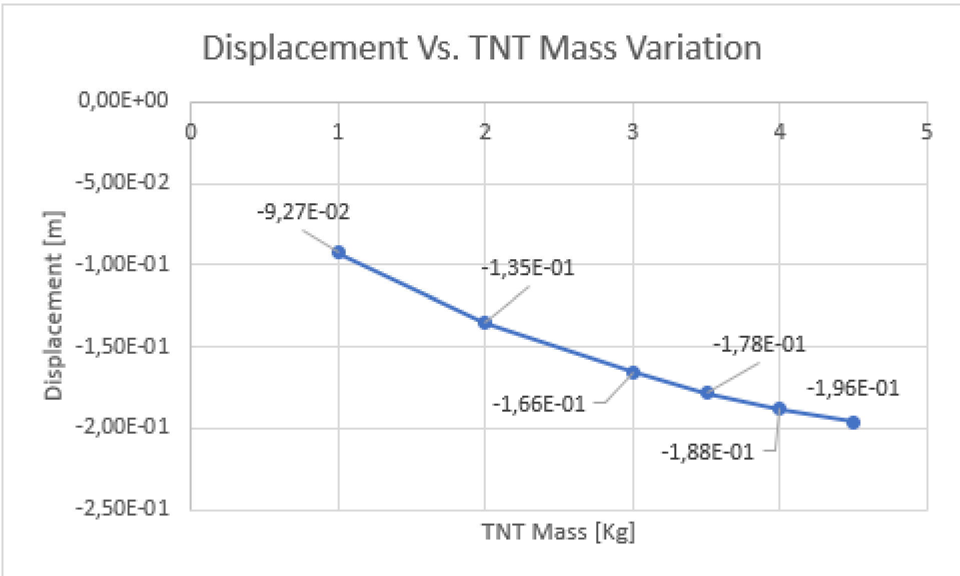


Fig. 4.25. Evolution of the displacements with TNT mass

5. CONCLUSION

After the analysis of the results, the conclusion obtained are explained in the following points:

- When it comes of select the most critical point of detonation for the integrity of the structure, the one coming from the hold of the aircraft has been tested as the one which produce the plastic deformation of the upper surface with less TNT mass. The main objective of the project has been achieved.

- In terms of the displacements of the target surface, they are very significant, not only in the simulation where the structure is plastic deformed, but also in the first simulation proposed with 1 [Kg] of TNT. These deformations can produce the collapse of the floor, causing damages to passengers or, for instance, the blast coming from the lateral of the aircraft can produce holes on the fuselage, the despressurization of the cabin and they can extract people to the outsite.

- Regarding the pressure field, it can be checked that there is a relation between the pressure field obtained and the results for stresses. Two types of pressure have been recorded: the compressive pressure in most of the cases located where the displacements and rotation are restricted, the boundary conditions, at the ends of the target surface. The plastic deformation has appeared in this section, as it has been shown in (4.18).

  Considering the tensile pressure field, it is predominant at the points where the upper floor joins to the frames, where the skin of the fuselage is reinforced.

  In general, the compressive pressures have been more critical compared to the tensile ones, as they have caused the plastic deformation of the structure.

5.1. Future Developments

Based on the numerical model implemented as well as all the results obtained, in future projects the following points could be implemented:

- The rest of the components taking part of the fuselage can be included in the analysis, due to the fact that these can also cause the failure of the structure.

- Although only three points have been selected for this project, there are many points that can be introduced in order to study the effect from a different view.

- Regarding the material used, due to the importance of the composites in the aeronautical insdustry nowadays, they can be implemented in the same way as the Aluminum or a mix of materials to obtain a more realistic model, compared to the real life.

- Other methods for modelling the blast can be tested is the ALE Method (Arbitrary Lagrangian-Eulerian Method) or the combination of CONWEP and ALE, as it has been explained in the State of the Art chapter.

- Finally, details in the aicraft fuselage can be introduced such as windows or door, but these type details will only cause an increment in stress concentrations, due to its geometry.

BIBLIOGRAPHY

[1] EFE, "Terroristas con implantes de explosivos dentro del cuerpo, la nueva amenaza", *La voz de Galicia*, 2011.

[2] I. Piquer, "Los pasajeros de un avión detienen a un terrorista con una bomba en el zapato", *El País*, 2001.

[3] (2018). Vuelo 203 de avianca, [Online]. Available: https://es.wikipedia.org/w/index.php?title=Vuelo_203_de_Avianca&oldid=109539028.

[4] J. M. Abad, "Así cambió el 11-s la forma de montar en un avión", *El País*, 2016.

[5] T. Vicente, "Así han cambiado las medidas de seguridad en los aviones tras el accidente de germanwings", *ABC*, 2016.

[6] A. S.-M. S. Alsimet. (2017). Usos del aluminio en la industria, [Online]. Available: http://alsimet.es/noticias/usos-del-aluminio-en-la-industria.

[7] A. Carnicero. (). Introducción al métodod de los elementos finitos, [Online]. Available: https://www.iit.comillas.edu/carnicero/Resistencia/Introduccion_al_MEF.pdf.

[8] M. Bermejo Castro, "Modelo de elementos finitos explícitos para explosiones en estructuras reticuladas de hormigón armado. aplicaciones al estudio del colapso de edificios", Tesis Doctoral, Universidad Politécnica de Madrid, 2015.

[9] (2018). Descripción general de abaqus, [Online]. Available: https://www.3ds.com/es/productos-y-servicios/simulia/productos/abaqus/.

[10] (2017). Skin (aeronautics), [Online]. Available: https://en.wikipedia.org/w/index.php?title=Skin_(aeronautics)&oldid=770777908.

[11] (2018). A350 xwb family, [Online]. Available: https://www.airbus.com/aircraft/passenger-aircraft/a350xwb-family.html.

[12] (2017). Shell vs. solids. finite element analysis quick review, [Online]. Available: https://www.linkedin.com/pulse/shells-vs-solids-finite-element-analysis-quick-review-kuusisto-p-e-/.

[13] M. Rodriguez Millán, "Análisis numérico del comportamiento frente a impacto de aluminio 2024-t351 sometido a ensayo de taylor", Proyecto de Fin de Carrera, Universidad Carlos III de Madrid, 2009.

[14] (1976). Journal of the mechanics and physics of solids, [Online]. Available: https://www.sciencedirect.com/.

[15] V. Sánchez Gálvez, "Tenacidad de la fractura dinámica", Universidad Politécnica de Madrid, Informe, 2008.

[16] (1969). Análisis explícito, [Online]. Available: http://sergioariasfernandez.com/analisis-explicito/.

[17] (2017). Límite elástico, [Online]. Available: https://es.wikipedia.org/w/index.php?title=L%C3%ADmite_el%C3%A1stico&oldid=97243185.

[18] (2010). Structural response of a blast loaded fuselage, [Online]. Available: https://www.researchgate.net/publication/267962737.

[19] J. Gilbert Kaufman, "Understanding the aluminum temper designation system", in *Introduction to Aluminum Alloys and Tempers*. 2000.

[20] M. C. Y. Niu. (1988). Airfracme structural design.

[21] A. Remennikov, "A review of methods for predicting bomb blast effects on buildings", University of Wollongong, Research Online, 2003.

[22] R. F. Danesi, "Experimental and computational analysis of plates under air blast loading", International Journal of Impact Engineering, PERGAMON, Research online, 2001.

[23] J. A. García Gómez, "Influencia del criterio de fallo en simulaciones númericas de impacto en aluminio", Trabajo de Fin de Grado, Universidad Carlos III de Madrid, 2016.

[24] Y. Uzquiano Díez, "Análisis numérico de la influencia de la geometría de un proyectil fsp impactando en una placa de kevlar 29", Trabajo de Fin de Grado, Universidad Carlos III de Madrid.

[25] J. Reboul Copra, "Modelización numérica del comportamiento de un material compuesto de tejido de fibra de carbono y matrix epoxi ante cargas dinámicas", Trabajo de Fin de Grado, Universidad Carlos III de Madrid, 2012.

[26] R. Ruíz-Castro Alcobendas, "Estudio paramétrico del proceso de corte ortogonal mediante el método de los elementos finitos", Proyecto de fin de Carrera III de Madrid, Universidad Carlos, 2009.

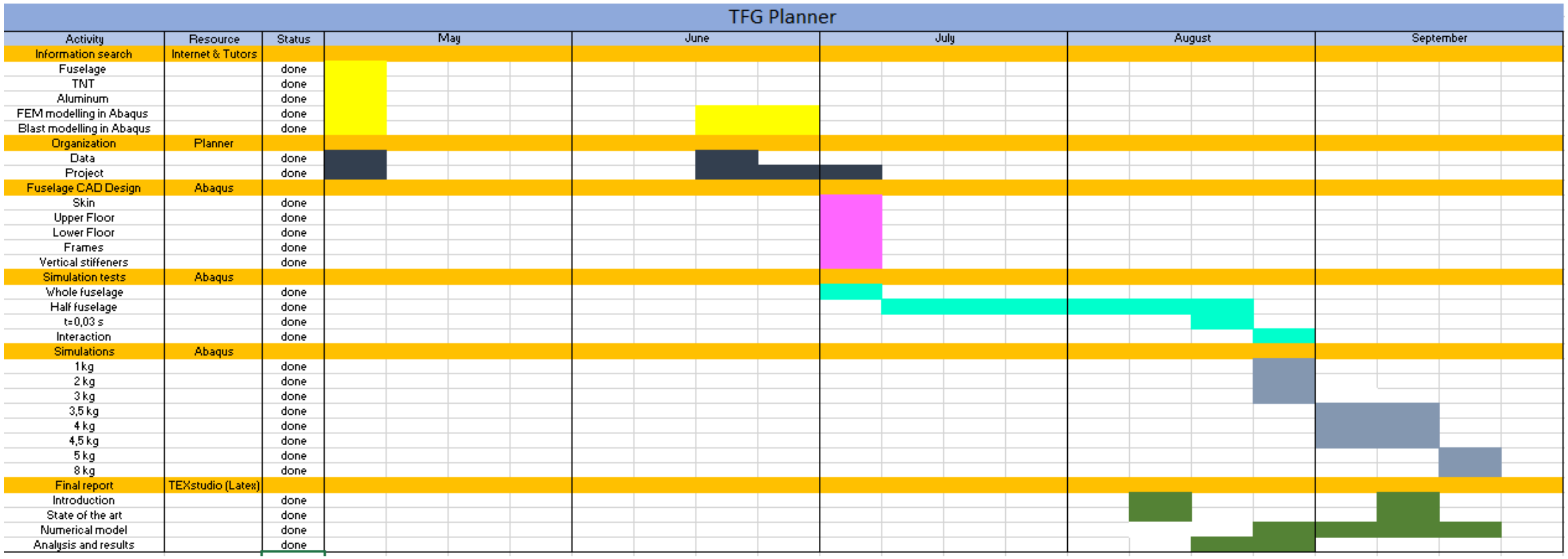
[27] *Norma une-en 13630*, 2003.

[28] *Norma une*, 2018.

ANEXO I: GANTT'S DIAGRAM

The color legend used for the Gantt's Diagram is:

Color legend	
	Information
	Organization
	CAD Design
	Simulation tests
	Simulations
	Final Report


TFG Planner
Activity
Resource
Status
May
June
July
August
September
Information search
Internet & Tutors
Fuselage
done
TNT
done
Aluminum
done
FEM modelling in Abaqus
done
Blast modelling in Abaqus
done
Organization
Planner
Data
done
Project
done
Fuselage CAD Design
Abaqus
Skin
done
Upper Floor
done
Lower Floor
done
Frames
done
Vertical stiffeners
done
Simulation tests
Abaqus
Whole fuselage
done
Half fuselage
done
t=0,03 s
done
Interaction
done
Simulations
Abaqus
1 kg
done
2 kg
done
3 kg
done
3,5 kg
done
4 kg
done
4,5 kg
done
5 kg
done
8 kg
done
Final report
TEXstudio (Latex)
Introduction
done
State of the art
done
Numerical model
done
Analysis and results
done

ANEXO II: SOCIO-ECONOMICAL IMPACT

One of the impacts that one of the situations explained in the Introduction Section of this project, from the point of view of the aeronautical industry is the continuous research of solutions to avoid the most damage in the structure and, so the least damage on people traveling inside the aircraft.

This research suppose huge money investments, by companies in the sector, for the reinforcement of the aeronautical structures and the creation of new materials capable of support not only variations in pressure or atmospheric conditions (normal configurations) but also either intentional and accidentally detonations of explosives.

Not only the security increment must be implement in aircrafts but also in airports, as they are the gates to reach the mean of tranport. In airports, very extrict security regulations were applied after suffered these events such as regulations for the transport of liquids inside the passengers' cabin, security controls to any kind of electronic devices like computers or tablets or check shoes and coats.

Also, the increment in security guards not only in the security controles named before but also walking around in terminals.

Regarding the social impact, this increment in security in airports has made people, that have to take an airplane, arrive minimum 2 hours in advance to the airport, wait in long qeues in the check-in counters in order to check in the luggage or going through the security controls named previously.

The most important impact that a terrorist attack can has on people, is the fear of travellin in this mean of transport or find a fogotten luggage in the middle of the airport. Therefore, it is important to make that fear disappear.

ANEXO III: BUDGET

Taking into account that when the project started, the extra information and learning about how to model a blast in the software *Abaqus* is nor included in the following Budget. It correponds to an aeronautical engineer with the enough knowledge to use this software simulation with any type of problem.

Budget

Name: Rocio Fontecha Ballester:
address: Calle Recaredo 10, 4 10 Sevilla
Phone: +34 639 45 39 39
E-mail: 100315599@alumnos.uc3

Client Data

Name	Universidad Carlos III de Madrid
Address	
Phone	

Budget date	24/09/2018	Validity:	

DESCRIPTION	HOURS	PRICE	% DTO.	PRICE DTO.	TOTAL
Information search	10,00	16,00		- €	160 €
Organization of the project and data	3,00	16,00		- €	48 €
Fuselage Model Design (CAD)	50,00	16,00		- €	800 €
Data introduction	15,00	16,00		- €	240 €
Simulation tests	150,00	16,00		- €	2.400 €
Final Simulations	50,00	16,00		- €	800 €
Tenchical Report of the simulation	7,00	16,00		- €	112 €

Gross Total		4.560 €
I.V.A. %	21%	957,60 €
Total Budget		**5.517,60 €**

Method of payment : Banking transfer

Signature of the person who makes the budget.	CCEPT THE BUDGET. Name, last name and signature of the clie